Part 2:

PowerFrame Reference Manual

© 2006, BuildSoft NV

All rights reserved. No part of this document may be reproduced or transmitted in any form or by any means, electronic or manual, for any purpose whatsoever, without the prior written consent from BuildSoft.

The programs described in this manual are subject to copyright by BuildSoft. They may only be used by the licensee and may only be copied for the purpose of creating a security copy. It is prohibited by law to copy them for any other purpose than the licensee's own use.

Although BuildSoft has tested the programs described in this manual and has reviewed this manual, they are delivered 'As Is', without any warranty as to their quality, performance, merchantability or fitness for any particular purpose. The entire risk as to the results and performance of the programs, and as to the information contained in the manual, lies with the end-user.

1 Contents

1 CONTENTS 3

2 INTRODUCTION 6

3 REFERENCE 7

3.1 WORK SPACE DESCRIPTION 7

3.1.1 Showing general parameters 7

3.1.2 Selecting elements 9

3.1.2.1 Directly on the screen 9

3.1.2.2 Using the menu 9

3.1.2.3 Using the 'Ctrl' or 'Alt'-button 11

3.1.2.4 Combined selections 11

3.1.3 Intelligent cursor 11

3.1.4 Zoom & pan 11

3.1.5 Hide/show selection 12

3.1.6 Material library 13

3.1.7 Cross-section library 14

3.1.8 The 'Geometry' window 16

3.1.8.1 The grid 16

3.1.8.2 Drawing plane 18

3.1.8.3 The geometry icon toolbox 19

3.1.8.3.1 Selection arrow 19

3.1.8.3.2 Draw bars 19

3.1.8.3.3 Remove bars 19

3.1.8.3.4 Divide bars 20

3.1.8.3.5 Intersection of members 20

3.1.8.3.6 Translation & copy 20

3.1.8.3.7 Rotation 21

3.1.8.3.8 Mirror 22

3.1.8.3.9 Extrusion 23

3.1.8.3.10 Pre-defined structures 24

3.1.8.3.11 Boundary conditions 24

3.1.8.3.12 Diaphragms 25

3.1.8.3.13 Rigid or hinged nodes 26

3.1.8.3.14 Connection at bar ends & tie rods 27

3.1.8.3.15 Constant and variable cross-sections based on cross-section types 29

3.1.8.3.16 Selecting a cross-section from the cross-section library 31

3.1.8.3.17 Link to Section Utility for general cross-section types 32

3.1.8.3.18 Orientation of cross-sections 32

3.1.8.3.19 Eccentricity of bars 32

3.1.8.3.20 Selecting materials from a library 33

3.1.8.3.21 Buckling and lateral buckling lengths 33

3.1.8.4 Moving bars and nodes 34

3.1.8.5 Modifying bars and nodes 35

3.1.8.6 Grouping and ungrouping bars 36

3.1.8.7 Element types 37

3.1.8.8 Copy/paste of boundary conditions & cross-sections 38

3.1.9 The 'Loads' window 38

3.1.9.1 The loads icon toolbox 38

3.1.9.2 Load cases 39

3.1.9.3 Load combinations 43

3.1.9.4 Defining loads 46

3.1.9.4.1 Loads at nodes 46

3.1.9.4.2 Loads on members 47

3.1.9.4.3 Temperature exposure 51

3.1.9.4.4 Pretensioning load 51

3.1.9.4.5 Generating climate loads 52

3.1.9.4.6 Surface loads 52

3.1.9.4.7 Modifying or removing loads....55
3.1.9.4.8 Copy/paste of loads....55
3.1.9.4.9 Dynamic mass....56
3.1.10 The 'Plot' window....56
3.1.10.1 Plot parameters....56
3.1.10.2 The plot icon toolbox....57
3.1.11 The 'Data' window....60
3.1.12 The 'Results' window....61
3.2 CALCULATION OF BUCKLING LENGTHS....62
3.3 DESIGN ANALYSIS....64
3.3.1 Static analysis....64
3.3.2 Global structural imperfections....68
3.3.3 Steel & timber design analysis....68
3.3.3.1 Selection of the design code....69
3.3.3.2 Steel design parameters....69
3.3.3.3 Timber design parameters....70
3.3.3.4 Verification of the cross-section strength....71
3.3.3.5 Verification of the buckling stability....72
3.3.3.6 Cross-section optimization....74
3.3.3.7 Loads histogram....77
3.3.4 Calculation of reinforcement quantities....78
3.3.4.1 Selection of R.C. design code....78
3.3.4.2 Concrete parameters....79
3.3.4.3 Reinforcement parameters....81
3.3.4.4 Organic calculations....82
3.3.5 Modal analysis....84
3.4 PRINTING MODEL DATA AND RESULTS....86
3.4.1 Printer configuration....86
3.4.2 Printing a window....87
3.4.3 Printing a report....87
3.4.3.1 Tab page 'General'....87
3.4.3.2 Tab page 'Geometry'....89
3.4.3.3 Tab page 'Loads'....90
3.4.3.4 Tab page 'Plot'....91
3.4.3.5 Tab page 'Data'....93
3.4.3.6 Tab page 'Results'....94
3.4.3.7 Additional options....97
3.4.3.7.1 Saving and reading printing preferences....97
3.4.3.7.2 Saving reports as RTF file....97
3.4.4 Print preview....98
3.5 SAVING AND OPENING PROJECTS....99
3.5.1 Saving a PowerFrame project....99
3.5.2 Opening a PowerFrame project....100
3.6 PREFERENCES....101
3.6.1 General parameters....101
3.6.2 Units and decimals....102
3.7 IMPORTING AND EXPORTING DATA....103
3.7.1 Import/export to DXF....104
3.7.2 Import/export to DSTV....104
3.7.3 Export to ConCrete Plus....104
3.7.4 Export to Microsoft Excel....105
4 CONNECTION DESIGN....106
4.1 DETAIL DESIGN OF CONNECTIONS....106

4.2 CONNECTION LIBRARY 112
4.3 VERIFICATION OF NODES & CONNECTIONS INSIDE POWERFRAME 115

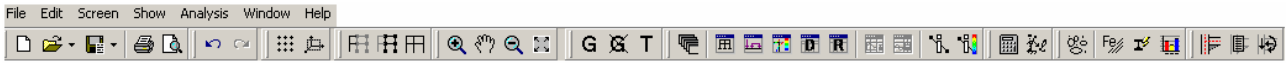
2 Introduction

This second part of the PowerFrame User's Manual provides more detailed information about the functions and procedures incorporated in PowerFrame, including a review of the implemented analysis strategies together with a more theoretical background. Above all, PowerFrame is and remains a design and analysis tool. Understanding and interpreting correctly the results of the analysis is the key to a successful and efficient use of the program. Accordingly, this manual remains also highly valuable for more experienced users.

3 Reference

3.1 Work space description

PowerFrame main window – which appears when activating the program - includes a menu bar and an icon toolbar as shown below.



This main window includes 5 working windows, displayed to the user in the following order :

- Untitled : Geometry
- Untitled : Loads (+ name of active load case)
- Untitled : Plot (+ name of presented results type)
- Untitled : Data
- Untitled : Results

To stack the default windows, use the menu function 'Window – Stack Windows' or the icon .

To access a window, the user can either select it directly or access it through the menu 'Window'.

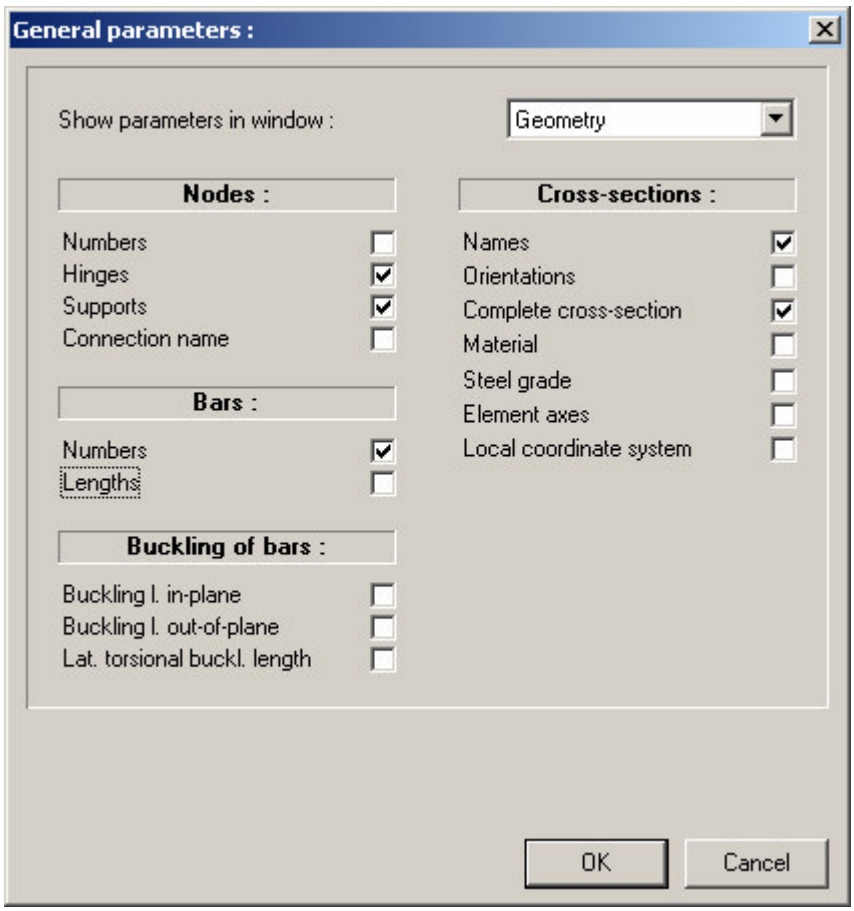
The graphical windows 'Geometry', 'Loads' and 'Plot' include an icon toolbox allowing a direct access to modeling or post-processing functions. The use of those toolboxes will be further discussed in the relevant sections of this manual.

To hide the toolbox, use the menu function 'Window – Icon toolbox'. The same operation is required to show the toolbox when hidden.

Prior to discuss each working window, a certain number of principles common to all graphical working windows will first be presented.

3.1.1 Showing general parameters

In order not to visually overload the working windows, the user can specify which model data are to be shown, by using the menu function 'Show – General parameters…', or by pressing in the icon toolbar.



A dialogue window will then appear as shown above, common to the 3 graphical working windows. The pull-down menu indicates the window to which the parameters actually relate. By default, this will be the window in the front.

Through the above dialogue window, the user will be able to visualize the following information about the 'Geometry' field:

- node numbers
- hinges
- supports
- name of connections (if included in the model)
- member numbers
- member lengths
- buckling length of members (in-plane and out-of-plane buckling)
- lateral torsional buckling length
- cross-section names
- cross-section orientation
- complete display of the cross-section on the screen
- type of materials
- steel grade – if applicable
- element axes

- local coordinate systems

All parameters are saved in order to remain valid for any future use of the software, until further modifications by the user.

3.1.2 Selecting elements

In order to assign specific properties to one or more elements of the model (either nodes or bar members), the user first selects the appropriate elements. Compared to other methods where the user has first to specify the properties to be defined and only then, select the elements to which these properties should be assigned, this procedure allows the user to work faster, as he can easily select multiple elements at a time, and also significantly reduces the risk of erroneous assignments, as the user has a direct visual feedback over the selection and the assignment of properties.

Several selection procedures are available to the user, as explained below.

3.1.2.1 Directly on the screen

Using the mouse, the user can select any element (node or member) on the screen by either clicking directly on the element to be selected or by drawing a selection field around the elements to be selected. To create such a selection field, use the left-hand button of the mouse, to define the upper left corner of the window. Keeping this button pressed, the user moves the mouse over the screen and will notice a rectangle appearing in dashed lines. Once the final mouse position (corresponding to the lower right corner of the selection window) is reached, the user releases the mouse button. All elements, nodes and members, located completely within the selection field are now selected.

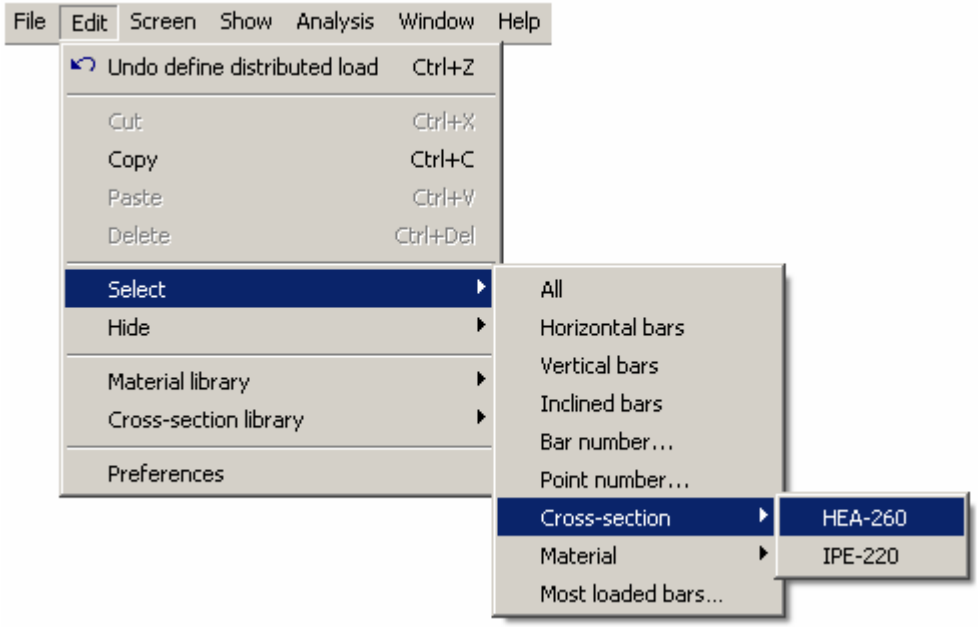
By performing the operation from the right to the left, all elements that are completely or partially located within the selection field will be selected.

To unselect the selected elements, a simple mouse click on the screen (making sure not to click on the geometry model) is sufficient.

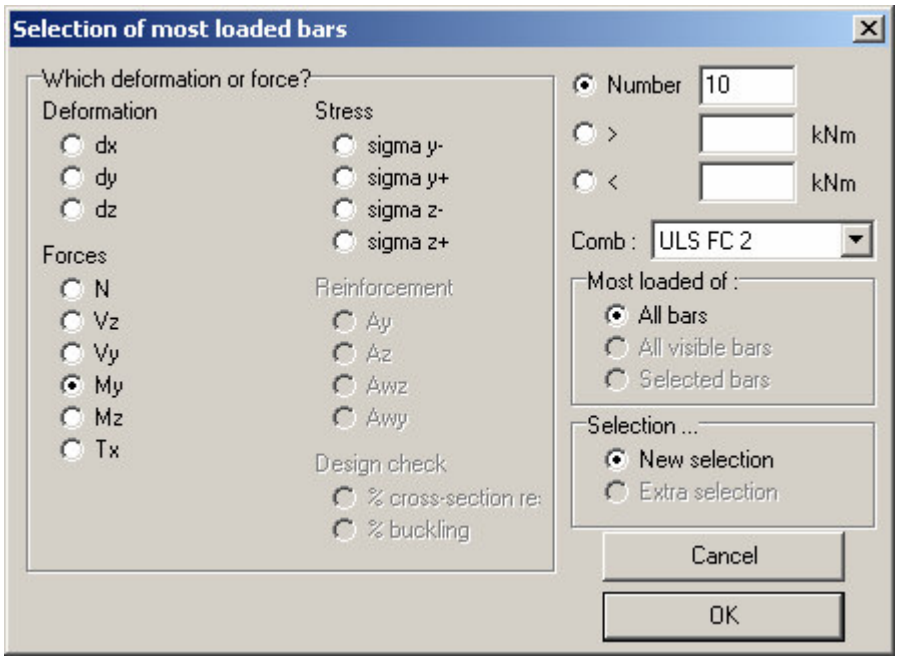
3.1.2.2 Using the menu

Alternatively, elements can be selected through the menu bar in accordance with a certain number of selection criteria. These criteria can include the specific properties (vertical/horizontal members, …), the cross-section characteristics, … as well as the loading type on the structural elements.

The figure below displays a number of selection criteria.



One of these criteria relates to the 'Most loaded members'. It only becomes active when the analysis has been performed by PowerFrame. When this selection criterion is chosen, a dialogue window will prompt the user to specify which results and load combinations should be considered.



For example, the user can ask PowerFrame to select those 10 members submitted to the maximum bending moment M_y under the load combination ULS FC 2 (example shown above).

3.1.2.3 Using the ‘Ctrl’ or ‘Alt’-button

As will be further explained, several structural elements can be combined into one specific ‘Type’. All elements belonging to one specific type can easily be selected simultaneously. Click the ‘Ctrl’-button and, while keeping it activated, use the mouse to choose all members to be selected. All members of the same type will now be selected and shown in bold on the screen.

PowerFrame also uses a group concept, to take into account the fact that several structural members actually correspond to one single physical element. This group concept will further be treated to full depth. For the moment, it is sufficient to know that a bar that has been divided into several sections remains known to PowerFrame as a single group. Selecting any element of a particular group by keeping the ‘Alt’-button pressed down will select the entire group.

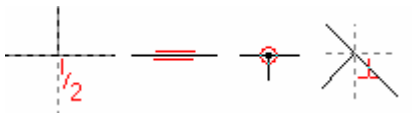
3.1.2.4 Combined selections

Several selection methods can easily be combined. For example, the user can make a selection of elements using any of the previously explained methods and then complete the selection using a different method or criterion. To make sure the current selection remains active, the user should keep the ‘Shift’-button pressed down.

3.1.3 Intelligent cursor

PowerFrame is equiped with an intelligent cursor able to automatically snap to specific points of interest. To check if this cursor can be used, the user will first verify whether the intelligent cursor has been activated. Use the menu ‘Edit – Preferences’ from the menu bar. In the dialogue window which appears, a section ‘Fly-over snap’ is available. To activate the latter, make sure the option ‘Use object snap’ is selected. A snap distance can be chosen by specifying the number of pixels.

The intelligent cursor is able to snap to members, to end nodes of members, to the mid-side nodes of members and to the orthogonal projection on members, as shown below.



3.1.4 Zoom & pan

To facilitate the use of the model, PowerFrame provides the user with ‘Zoom in’ and ‘Zoom out’ functions through the icons and in the icon toolbar.

To use the 'Zoom in' function, use the icon and then define the zoom window by drawing it directly on the screen. To zoom out, use the icon.

The pan-function allows the model geometry to be moved throughout the screen using the mouse. Push the button with the left-hand button of the mouse. Then, keeping the mouse button pressed down, move the mouse to see the model moving throughout the screen.

To fit the complete model to the working window, use the icon of the toolbar.

All above-mentioned functions are also available through the menus. Go to 'Screen' and the four top entries will give access to these model manipulation possibilities. Alternatively, the following shortcuts can also be used:

F10 : Zoom in
F11 : Zoom out
F12 : Fit to window

Finally, a last method is available for zoom in & out. Having clicked the right-hand button of the mouse, move it over the window, keeping the button pressed down. Moving the mouse up will zoom in on the model. Moving it down will zoom out. The zoom operation is always applied from the originally selected point which accordingly will remain visible at all times.

3.1.5 Hide/show selection

The graphical interaction with the geometry model can be facilitated as PowerFrame offers the possibility to mirror only part of the model geometry. Working on a partially hidden model will further facilitate the selection of elements as the hidden parts of the model will not be unnecessarily available for selection.

First, select the members that are to remain visible and those that are to be hidden. Next, if the selected members only are to be displayed, the icon should be activated. If, on the other hand, the selected bars are to be hidden, click on .

Using the third icon ( ) will make all members visible again.

To switch from visible to hidden members, select the complete range of visible members. Then click on first to visualize also the hidden members, and then continue directly with to hide the members previously selected.

3.1.6 Material library

PowerFrame includes a material library containing 3 default types of material: steel, timber and concrete, each material with its specific pre-determined properties. The user can, for instance, create different types of concrete, each one with its own characteristics. It is also possible to add other materials, for instance aluminum. In case a material is defined as being concrete, timber or steel, PowerFrame will use the relevant design Standards when performing the analysis. For all other types of materials, PowerFrame will always perform a complete elastic analysis and will deliver deflections, internal forces, stresses, … with the exception of additional design code checks.

The material library can be managed as follows:

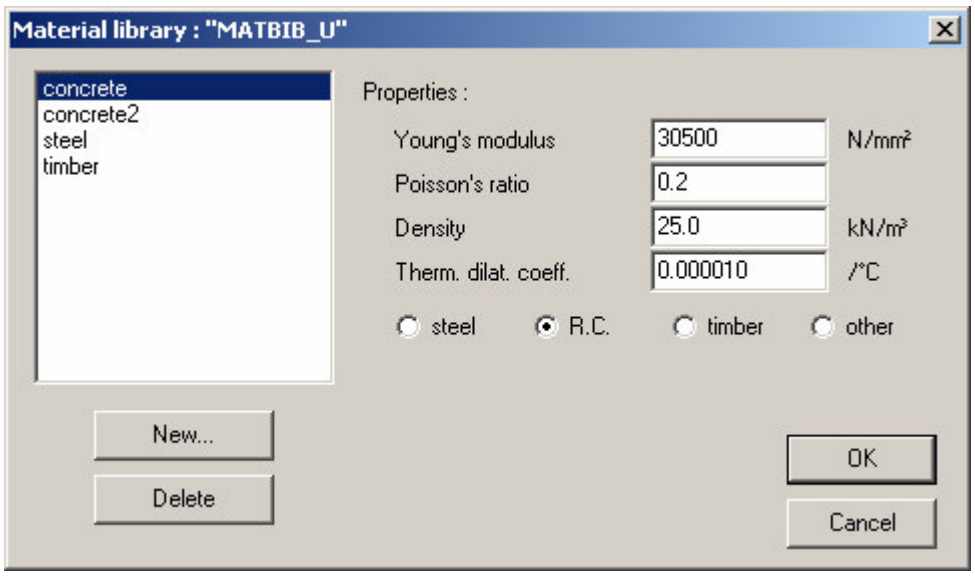
from the main menu, access the function 'Edit – Material library'.
Three operations are available:

- New…
- Select…
- Modify 'Matbib.efm'…

→The first entry allows to create a completely new material library. Once this library has been set up by specifying the name of a file where the material property definitions must be saved, it will automatically become active in PowerFrame. To actually define the contents of the library, the user should go through the 'Modify' menu entry.

→The second entry allows to select an existing material library as the active library in PowerFrame.

→The third entry gives the possibility to modify the contents of the active library. This will be done through the following dialogue field.



To change the properties of an existing material, select the name of the material at the left-hand side and simply change the values on the right-hand side. To introduce a new material, use the button 'New…'. Use the button 'Delete' to wip out a material from the list.

The four material characteristics required by PowerFrame to perform an elastic analysis, are:

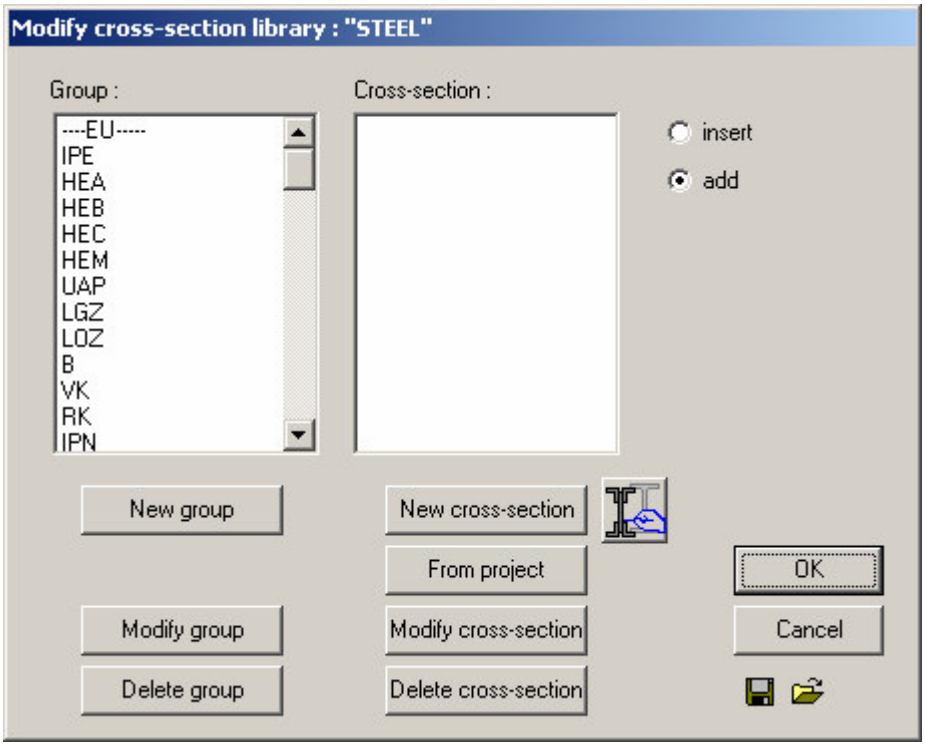
- Young's modulus ;
- Poisson's ratio ;
- density ;
- thermal dilatation coefficient.

Important note: It is recommended NOT to modify the default material library delivered with the PowerFrame installation, as this library is overwritten when installing an upgrade or update. If the default material library is to be modified, it is recommended to create a copy of the default library and then select this copy as the active material library. Within this library, the user can freely introduce changes and new entries without the risk of overwriting this information during future installation of updates or upgrades.

3.1.7 Cross-section library

PowerFrame enables to work with cross-section libraries. At the time of installation, PowerFrame is set up with a steel profile library, containing the standard European profiles, and with a timber cross-section library, with the most common cross-sections. Both libraries can easily be modified and extended. Additionally, the user can also create his own libraries.

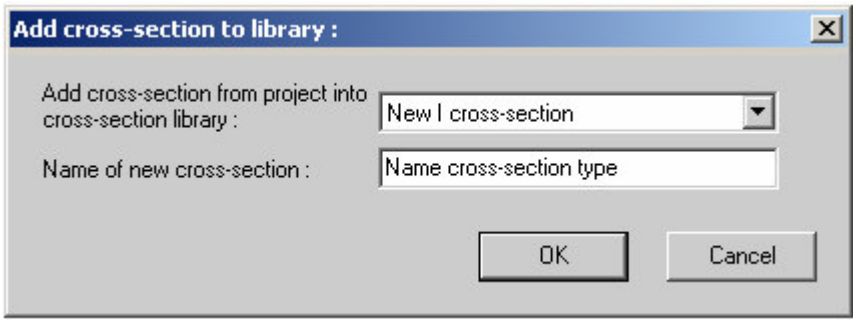
From the main menu, use the entry ‘Edit – Cross-section library’ to create a new library, select another library or modify the active library. Choosing ‘Modify…’, the following dialogue window will appear on the screen.



The left-hand column allows to select/define a specific group of cross-sections, while the right-hand column will further detail it through the selection/definition of a specific size.

Assume the user would like to define an entirely new group of profiles. ‘New group’ should first be selected, and the group name specified. Then the selection of this new group will be ensured from the list at the left-hand side. The ‘New cross-section’ button will then allow to specify the name, type and dimensions of the new profile. The same operation will have to be repeated for each new cross-section of the group.

The user will also notice a button named ‘From project’. This button gives access to a list of cross-sections which have already been defined in the active project but have not yet been included in the active cross-section library. Now these cross-sections should be introduced into the active library.



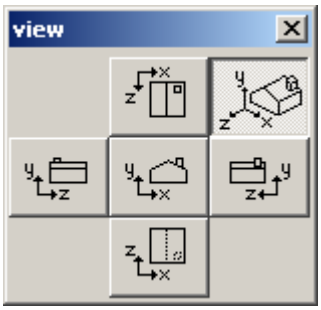
From now on, it will be possible to use the newly added cross-section in any PowerFrame project.

Finally, it should be noted that by selecting 'insert' on the right-hand side, a new group or a new cross-section will be inserted among the existing ones. In case 'add' is selected, PowerFrame will add the new group or new cross-sections at the bottom of the list.

3.1.8 The 'Geometry' window

The 'Geometry' window displays a graphical view of the model data. In this window, the model geometry can actually be drawn, cross-section properties assigned to members, boundary conditions assigned to nodes, and other specific properties related to nodes or members be specified.

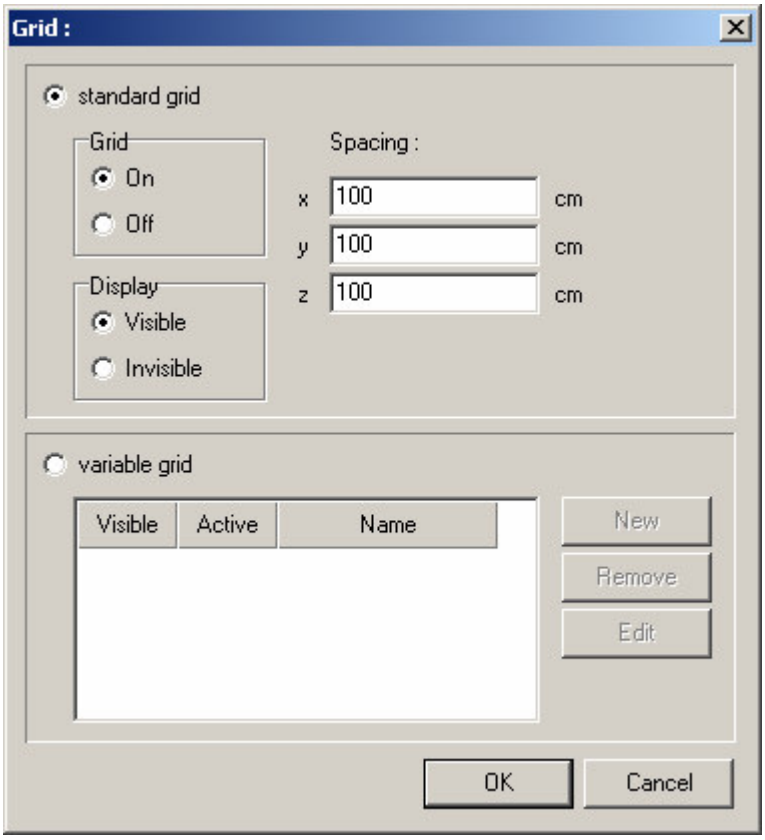
The modeling possibilities of PowerFrame will now be investigated, provided the 'Geometry' window is active. First, the user should notice two particular buttons in the lower left corner of this window. The 'View'-button allows the selection of a pre-defined view on the model. The other button allows to switch to a rendered visualization of the model.



3.1.8.1 The grid

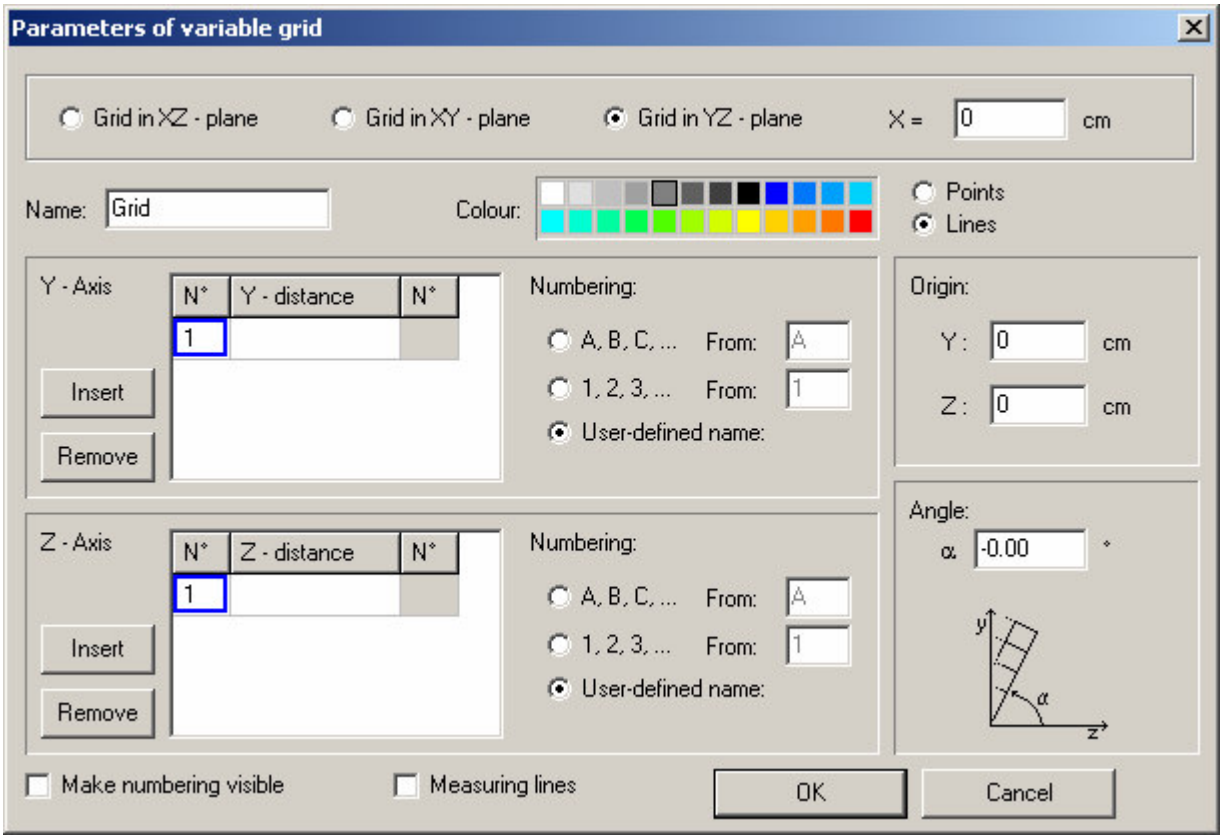
To facilitate the creation of a model geometry, PowerFrame allows the user to work on a grid. To visualize this grid in the working window, the user enters the menu 'Screen – Grid settings…', or clicks directly on the icon, which

will give access to a dialogue window enabling the grid settings to be specified (either regular or variable).



When the grid is defined as regular, the user can further specify whether this grid should be active, visible or invisible. In the window above, the grid resolution has been fixed at 100 cm in all 3 directions.

If the grid is defined to be variable, its resolution can be determined independently in 3 directions. Just select the button ‘New’ to create the variable grid definition, and then select the button ‘Edit’, which will give access to the dialogue menu shown below.



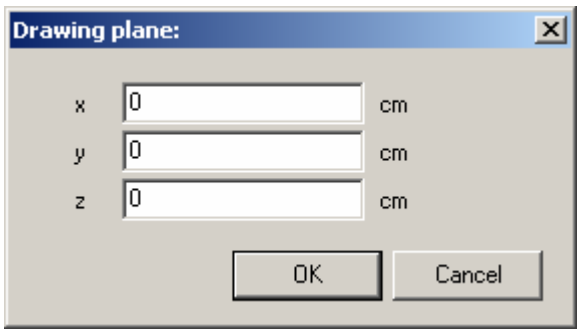
The first choice to be made relates to the plane for which the grid will first be defined (XZ, XY or YZ + distance to base plane). Then, the grid name and the colour should be specified, and whether it is to be visualized by points or lines.

Next, the user is to specify the distance between the grid axes. PowerFrame allows to complete the grid axes with a dedicated annotation, which should also be specified. To make it visible, the option 'Make numbering visible' at the bottom of the dialogue window should be selected.

The user can specify any number of grids and have them all visible at the same time. However, one grid only can be activated at the same time.

3.1.8.2 Drawing plane

When selecting a 2D view of the model geometry, the drawing plane will contain the origin of the global X/Y/Z coordinates system. It is possible to change the position of the drawing plane any time. Click on or use the menu entry 'Screen – Drawing plane…' to open the dialogue window below :

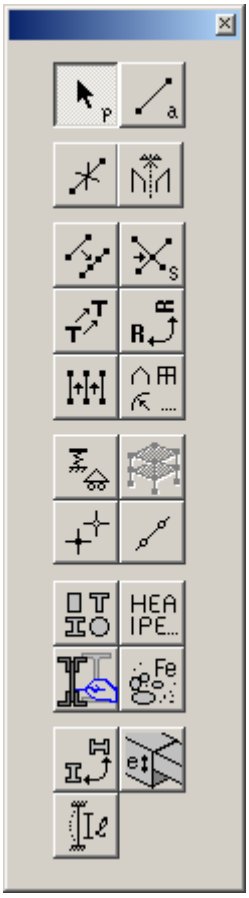


This dialogue menu allows to specify a point where the position of the drawing planes is to be located.

3.1.8.3 The geometry icon toolbox

The icon toolbox groups a complete set of modeling functions in a compact area on the screen.

3.1.8.3.1 Selection arrow



Using de-activates any other function active at a given moment. By clicking on this button, the cursor will come back to its original shape and the selection of any entity (node, member, …) will now be possible, within the 'Geometry' window.

3.1.8.3.2 Draw bars

allows to draw bar elements directly on the 'Geometry' window. Click on the icon and select a first point using the mouse. Move the mouse to the position of the second point, keeping the left-hand mouse button pressed down. Release the mouse button when the second point has been defined. A line will automatically be drawn between both points. Performing this operation using a 2D view, will enable the user to draw a line between 2 arbitrary points in the drawing plane. However, performing this operation using a 3D view, will only allow to draw new members between already existing members.

3.1.8.3.3 Remove bars

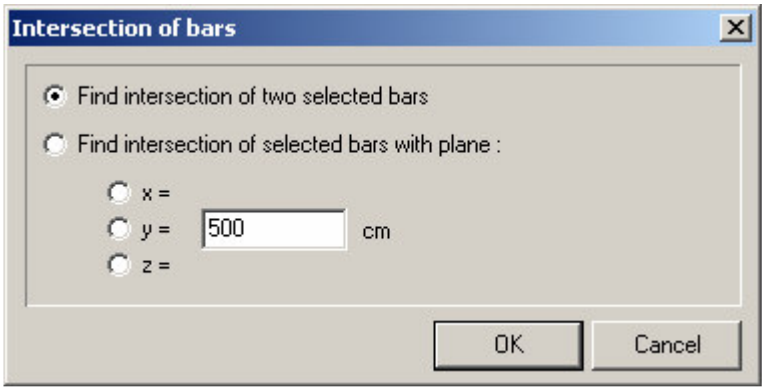
Using , will allow to easily wip out all members included in the current selection. Using the button ‘Delete’ or ‘Backspace’ will give the same result.

3.1.8.3.4 Divide bars

If the user wants to further divide selected members into multiple member elements, he will have to click on . A dialogue window will prompt the user to specify the number of divisions along the selected members.

3.1.8.3.5 Intersection of members

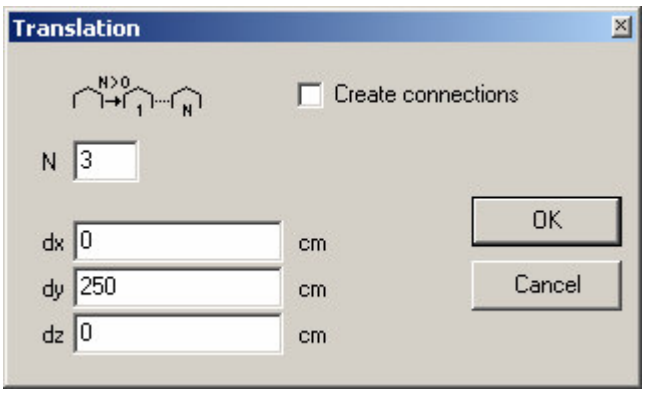
Should the user need to specify a node common to 2 intersecting members or a node at the intersection of a line and a plane, he should click on .



To appreciate the importance of this function, the user should bear in mind that 2 intersecting bars are not necessarily connected at the point of intersection, **unless a node is created explicitly at the point of intersection.**

3.1.8.3.6 Translation & copy

The button can be used to translate or copy selected members.



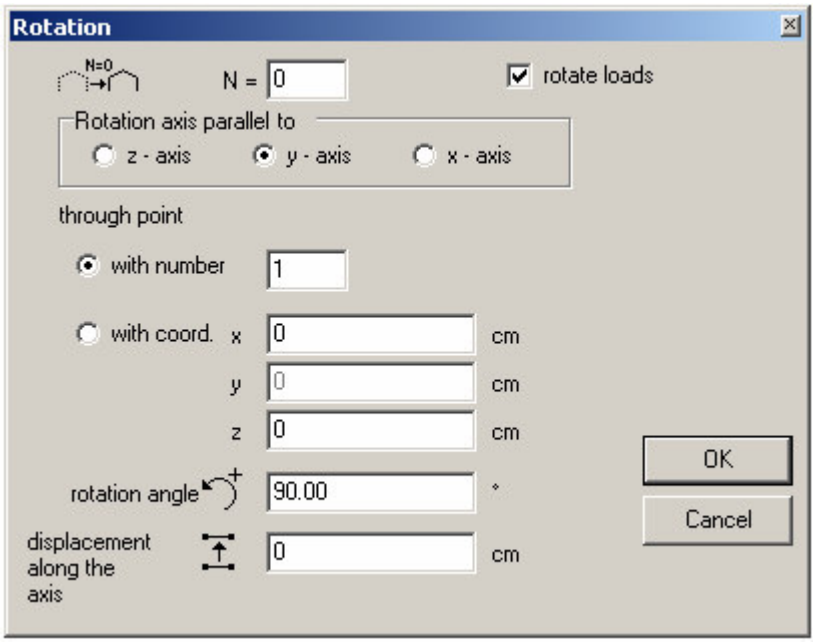
A dialogue menu will ask the user to specify the number of copies to be made (N). If a translation only is required, then N remains equal to 0, otherwise the user should simply specify the number of copies to be made, and next, define the translation vector to be used. If this is a copy operation, an optional request can automatically create members between the corresponding nodes of the original and the copied members.

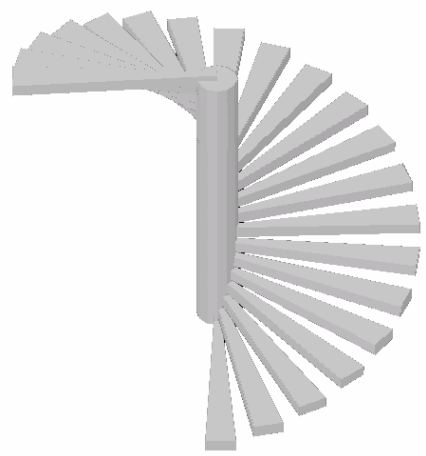
Note : Translation operations can easily be defined by drawing the translation vector directly between 2 existing nodes on the 'Geometry' window.

3.1.8.3.7 Rotation

Performing a rotation on (part of) the model is done following the same procedure as for the copy/translate operations. It is necessary to provide additional input parameters, such as the plane in which the rotation is to be performed, the rotation centre, the rotation angle and the number of copies to be made.

All these parameters can be defined through the dialogue file below, which is activated by clicking on the button .





For loads which have been defined relative to the global coordinate system of the structural model, direction and orientation relative to this global coordinate system will not be changed during a rotation operation. However, by selecting the option ‘rotate loads’, those loads will be locked to the corresponding bars during the rotation operation and so will be rotated along with those bars.

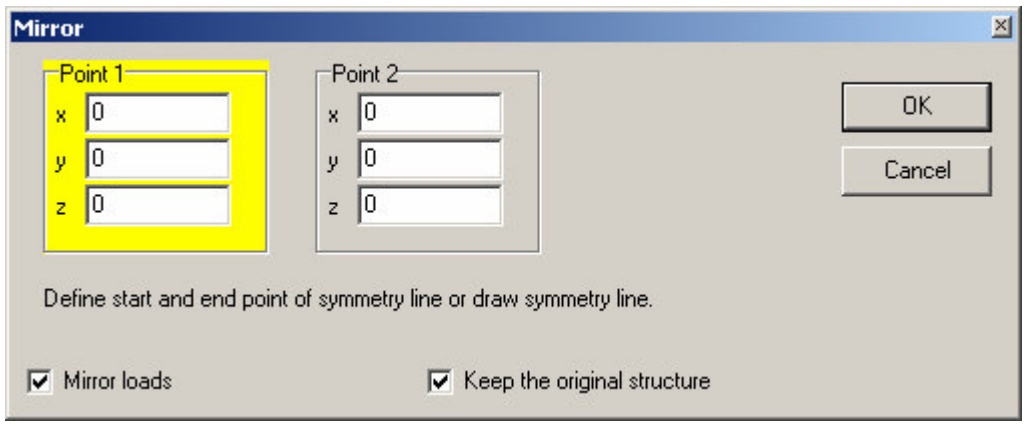
Loads which have defined relative to the local coordinate system associated to each bar element, will always be rotated along with the corresponding bar (whether or not the option ‘rotate loads’ has been activated.

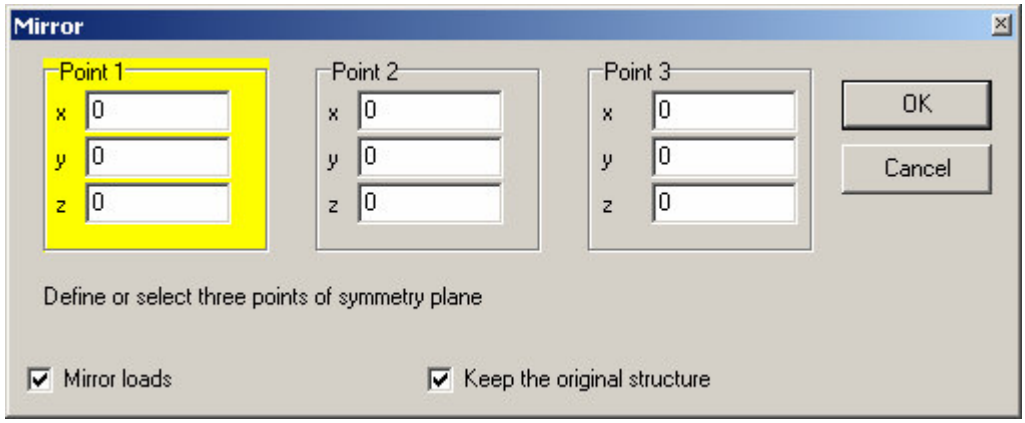
The rotation centre can also be defined by selecting a specific node in the ‘Geometry’ window. It is also possible to combine translation and rotation operations to create, for example, a staircase model as shown above.

3.1.8.3.8 Mirror

The icon allows to mirror selected members. Depending on the active view angle, the dialogue menu will ask for a symmetry line or a symmetry plane. A symmetry line (plane) is defined by

- ✓ either entering the coordinates of two (three) points belonging to the symmetry line (plane)
- ✓ or by drawing the symmetry line
- ✓ or by selecting three existing nodes of the symmetry plane directly in the ‘Geometry’ window.





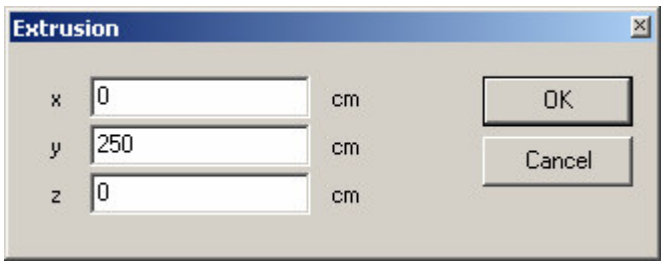
For loads which have been defined relative to the global coordinate system of the structural model, direction and orientation relative to this global coordinate system will not be changed during a mirror operation. However, by selecting the option ‘mirror loads’, those loads will be locked to the corresponding bars during the mirror operation and so will be mirrored along with those bars.

Loads which have defined relative to the local coordinate system associated to each bar element, will always be mirrored along with the corresponding bar (whether or not the option ‘mirror loads’ has been activated.

Finally, the option ‘Keep the original structure’ allows to maintain or reject the original structure as part of the modified model.

3.1.8.3.9 Extrusion

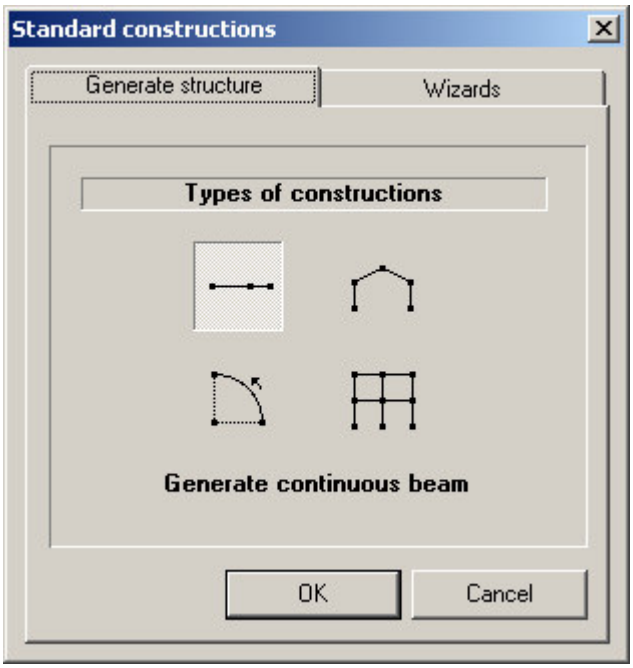
PowerFrame allows to extrude members from selected points in any given direction. Having selected the points, the user should click on and fill out the dialogue window below to specify the extrusion vector.

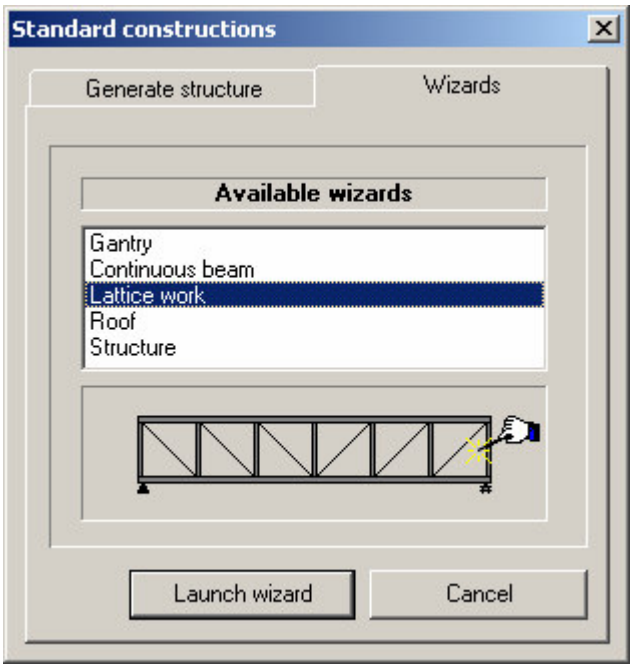


Note : The extrusion vector can also be defined by drawing it directly between 2 existing nodes on the 'Geometry' window.

3.1.8.3.10 Pre-defined structures

PowerFrame contains a library of typical structures, such as frames, spatial trusses, continuous beams, arches, etc. To access this library, click on the wizard icon .





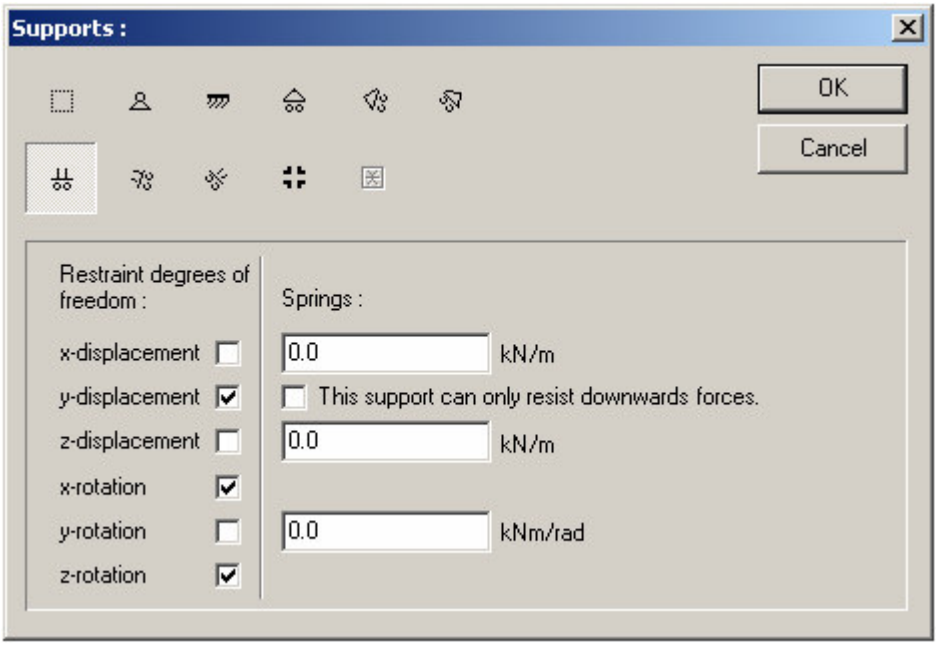
The window 'Generate structure' allows continuous beams, frames, arches and spatial trusses to be defined. The user has just to select the type of structure he wants to define and PowerFrame will ask to specify all data required to build the model. The second window, quite similar, gives access to other types of structures and allows a more detailed definition.

3.1.8.3.11 Boundary conditions

As a 3D analysis program, PowerFrame assigns six degrees of freedom (DOFs) to all nodes, 3 translational DOFs and 3 rotational DOFs. To enable a

correct analysis of any structural model, it is important to specify correctly which DOFs need to be constrainted at the supports of the structure.

To define the boundary conditions, click on , having previously checked that the nodes to which the user wants to associate a specific set of boundary conditions have already been selected.



In the dialogue window that appears, the user has a direct access to a number of pre-defined boundary conditions. By choosing any of these, the information on which DOFs are constrained is automatically displayed within the box.

Whenever required, this information can further be refined or customized by releasing or constraining any DOF from the list. In addition, translational or rotational stiffness values can be specified to model the actual stiffness of connections to the outside constructions. Finally, any boundary condition can be specified to be active only in a specific direction along a given axis. This allows for non-linear boundary conditions, dealing with compressive forces, but not tensile forces.

3.1.8.3.12 Diaphragms

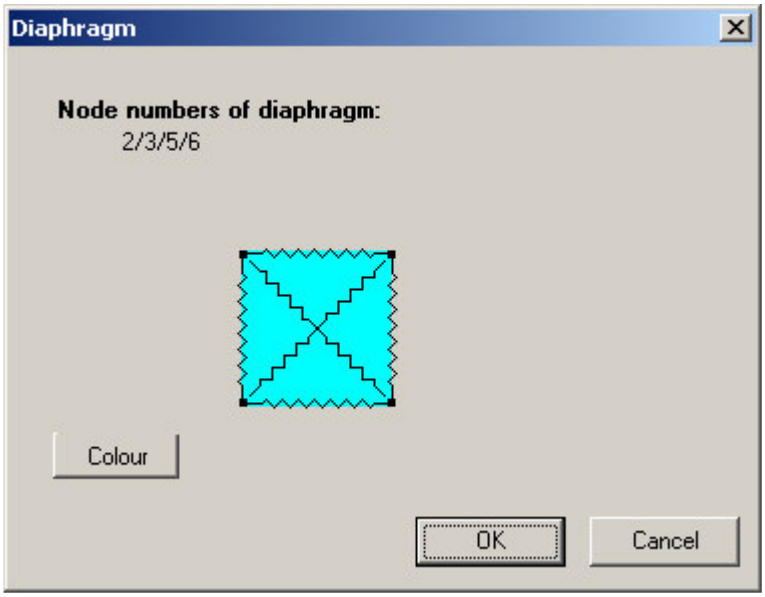
Diaphragms can be included in PowerFrame to model the effect of floor and wall panels on the deformations of the frame structure. All nodes belonging to the diaphragm will move as a rigid body within the plane of the diaphragm.

More specifically, a diaphragm will introduce very stiff springs between the selected nodes only for the DOFs relevant to displacements in the diaphragm

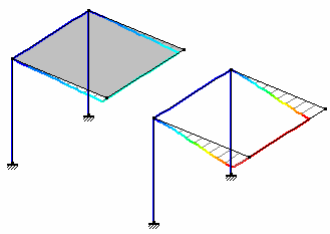
plane, but will not introduce any (bending) stiffness relative to displacements perpendicular to this diaphragm plane.

To define a diaphragm, the user will first select the nodes to be connected through a diaphragm by clicking on the icon. It should be noted that a minimum of 3 nodes - not aligned along one single axis – is required to determine the plane.

A window will be displayed confirming which nodes have been specified in the diaphragm definition. Click OK to confirm the definition.



The figure below shows the deformation of 2 similar structural models, subject to the same loads. The first model includes a diaphragm, the second one not.

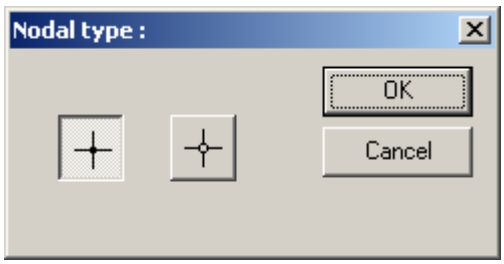


To delete a diaphragm from a model, select it with the mouse and use the 'Delete'-button or 'Backspace'-button ← to remove it.

3.1.8.3.13 Rigid or hinged nodes

By default, the nodes of a PowerFrame model will be rigid. However, any node can be specified as completely hinged (with respect to the strong AND

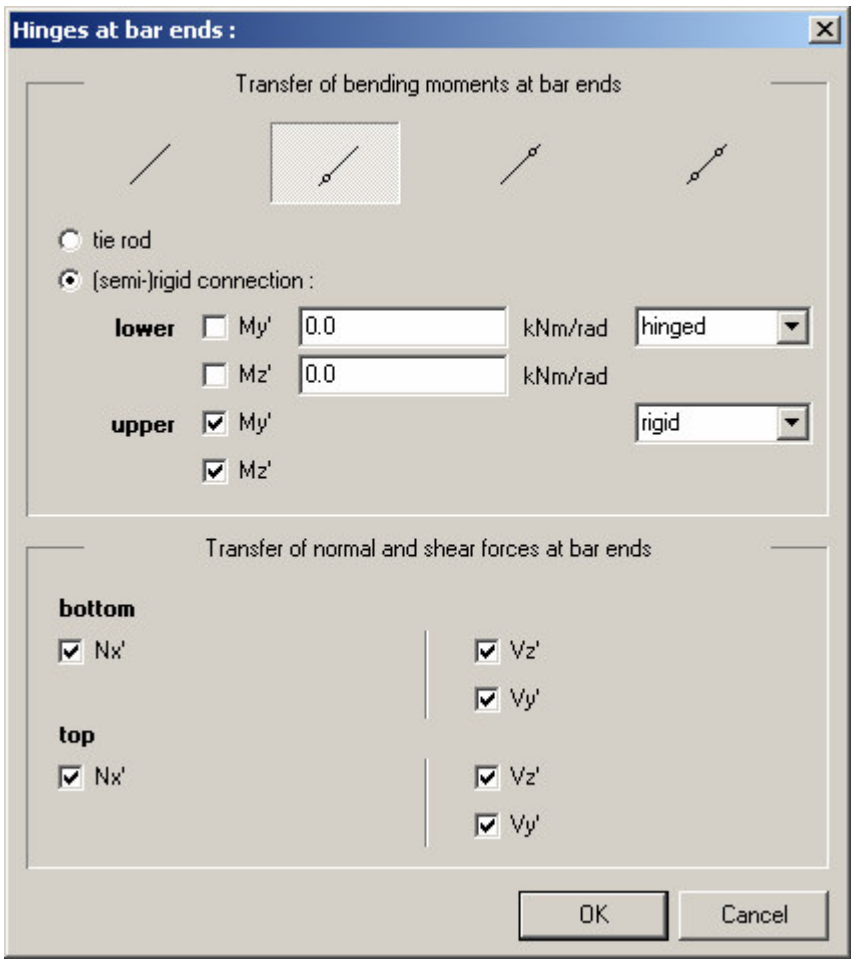
the weak axis of the cross-section) by selecting the considered node and then clicking on the icon .



Note: This function is to be used only if all bars meeting at a given node are tied to that node through hinges. In case only some bars have an hinged connection or if the hinged connection applies only to the weak OR the strong axis of the cross-sections, the node should be defined as rigid while the connection properties should be defined on the individual bars (see next section of this reference manual).

3.1.8.3.14 Connection at bar ends & tie rods

Quite often, connections between members of a steel frame are not really rigid. Therefore, PowerFrame can handle specific stiffness properties for any connection in a structural frame model. The user selects the bars involved and clicks on to open the dialogue window below.

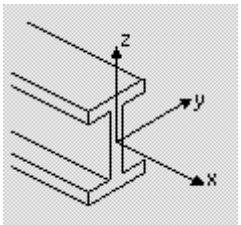


A first possibility offered by PowerFrame is the definition of selected members as tie rods. Members specified as tie rods cannot resist bending moments. Furthermore, the axial forces in the rods can only be tensile forces, which is ensured by PowerFrame through an iterative analysis, for all load combinations, which eliminates all tie rods working in compression. It should be noted that bars removed from the model can be different for each load combination.

Alternatively, members can be defined with semi-rigid connections at one or both ends. In this case, PowerFrame asks the user to specify the rotational stiffness of each connection.

For example, in the case where 2 bars (eg. a beam and a column) are tied through a semi-rigid joint, bending moments are transmitted from beam to column in function of a specific rigidity associated with one of the local bending axes, eg. « My' ». This is for instance the case when the joint stiffness has been calculated with PowerConnect. Another example, where a beam can move on top of a column may be modeled in PowerFrame by unselecting the appropriate « Vz' » in the dialogue window above, thus specifying that the connection cannot resist a shear force.

Note: y' and z' always refer to the local axes of the bar elements (see figure below).



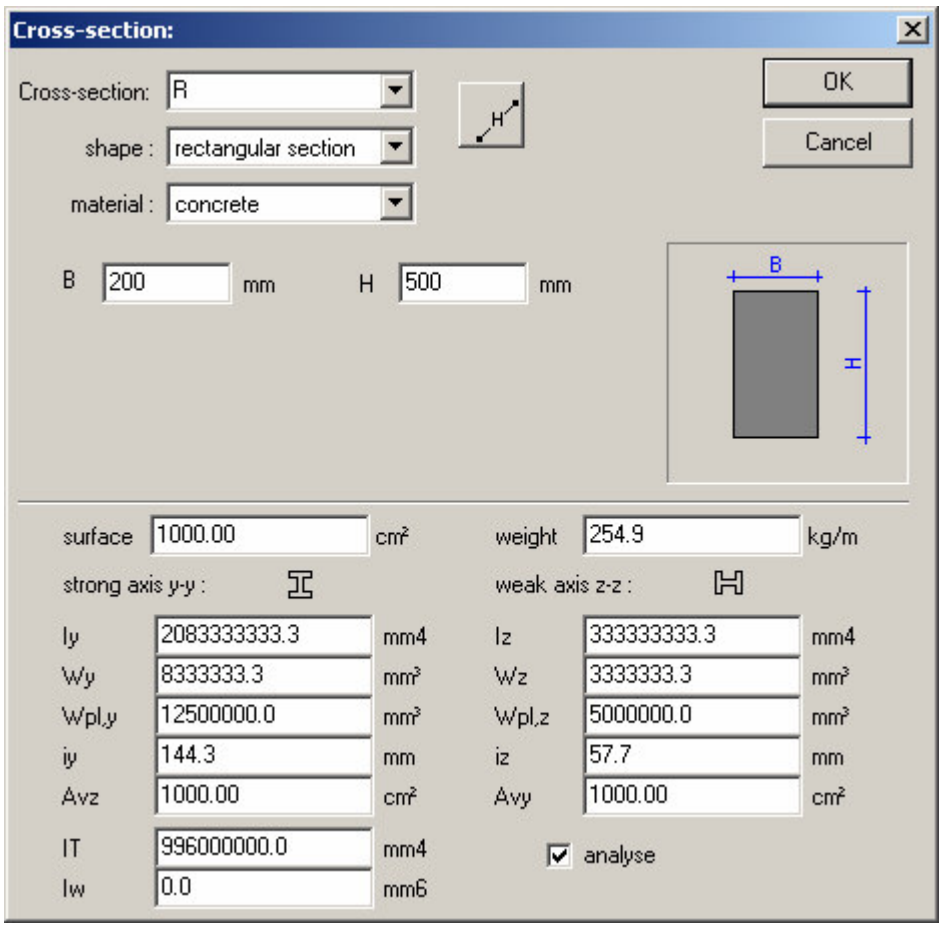
Note : the definition of the local axes orientation can be changed through the menu entry "Screen" – "Local coordinate system".

3.1.8.3.15 Constant and variable cross-sections based on cross-section types

Cross-sections can be defined in 3 different ways with PowerFrame:

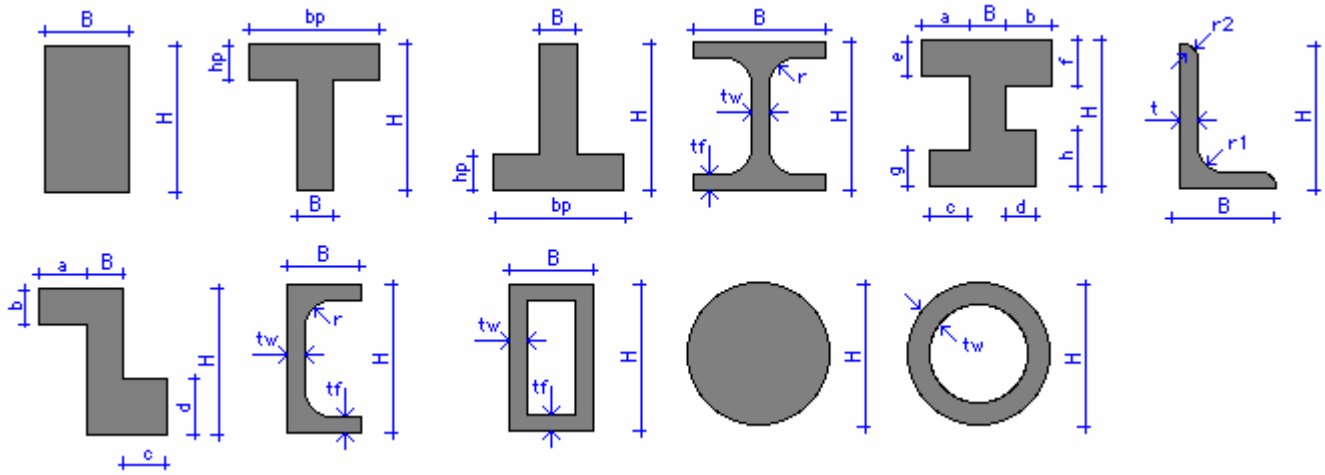
- by choosing the cross-section shape and specifying the dimensions,
- by selecting a cross-section from the cross-section library,
- by designing an arbitrary cross-section using SectionUtility.

When using the first method, the user selects the appropriate member(s) and clicks on to launch the related dialogue window.



He should then first give a name to the new cross-section to be defined.

Note: If there is already a series of cross-sections defined within the analysis project and if the user wants to assign any of those properties to a new member, he can use the pull-down menu inside the dialogue to select the required cross-section.



Next, a cross-section shape is to be selected from the PowerFrame library, together with a material from the material library. For more details on materials, please refer to the relevant section of this Manual.

Depending on the selected type of cross-section, an image will be displayed allowing to specify the dimensions required to fully characterize the cross-section.

To define a member having a variable depth, the user clicks on

to define H_1 and H_2 at both ends of the member. It will be noticed that the button will change automatically into

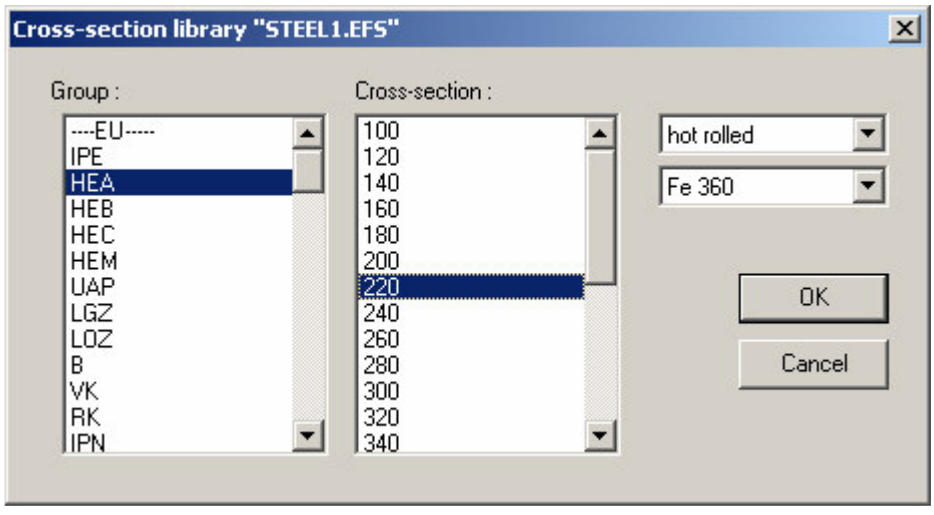
.

Cross-section characteristics are automatically calculated on the basis of the individual dimensions, provided the field 'Analyse' at the bottom of the window has been selected.

3.1.8.3.16 Selecting a cross-section from the cross-section library

The PowerFrame package is provided with a library of the most commonly used steel & timber profiles. The steel profile default library is active. To modify the active library or to select another library, refer to the relevant section of this Manual.
The selection of a cross-section from the active library will now be considered. By clicking on

, a dialogue window appears which allows the selection of a cross-section.



The first column displays the various families of cross-sections available in the library. The second column shows all available profiles within the selected family.
When using the library of steel profiles, the steel grade and production method (hot-rolled, cold-formed or welded) can also be specified. Both

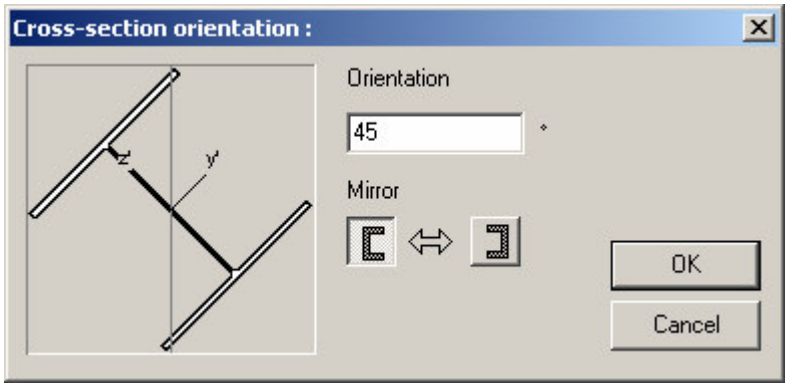
parameters are important when computing the profile strength and the possible buckling.

3.1.8.3.17 Link to Section Utility for general cross-section types

PowerFrame is enriched with a so-called Section Utility, allowing the user to draw any type of cross-section and to compute its characteristics. Having checked that all members to which the new cross-section should be assigned have been selected, click on to activate Section Utility. A specific chapter of the PowerFrame manual deals with Section Utility. This manual gives an overview of all functions offered by Section Utility, and further illustrates its use by means of a series of examples.

3.1.8.3.18 Orientation of cross-sections

Any cross-section selected from an existing cross-section library, has a default orientation in 3D space. If this orientation does not correspond to the desired orientation, PowerFrame always allows to change section orientation through the icon.

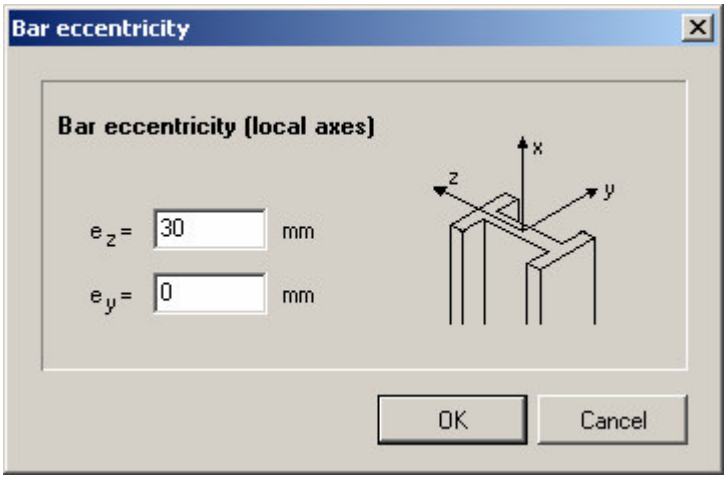


The left-hand side of the screen presents a view of the actual orientation of the cross-section at a given time. Just select the drawing and use the mouse to rotate it, or directly define the orientation angle by its numerical value. Two dedicated buttons also enable to mirror the cross-section.

3.1.8.3.19 Eccentricity of bars

During the analysis, PowerFrame can take into account any eccentricity existing between members meeting at a given node, thereby introducing secondary forces into the analysis related to the eccentricity. To include and define an eccentricity between members into the analysis, the icon

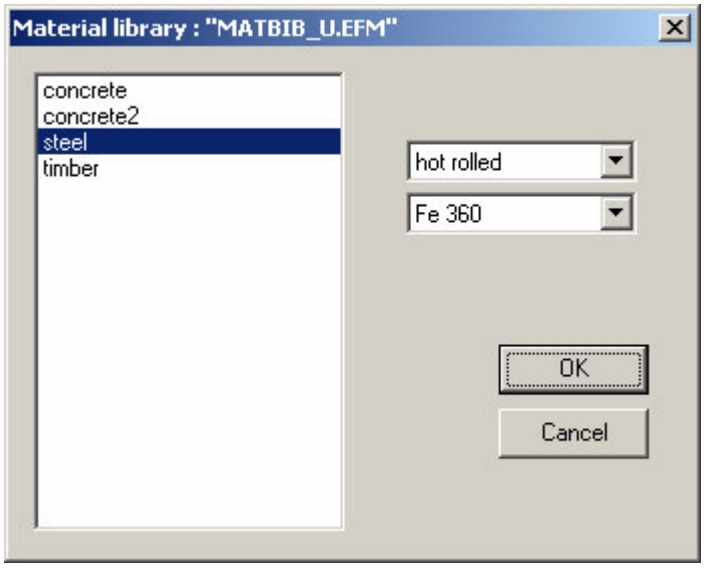
should be used. Eccentricities can be defined along both principal axes of the cross-section.



3.1.8.3.20 Selecting materials from a library

A material library is included in the total package of PowerFrame. Upon installation, this library contains 3 materials: steel, concrete and timber. The user can, any time, complete or modify this material library. How modifications can be made is explained in the relevant section of this manual. To assign a specific material from the library to a selected member, access the material library through the icon

.



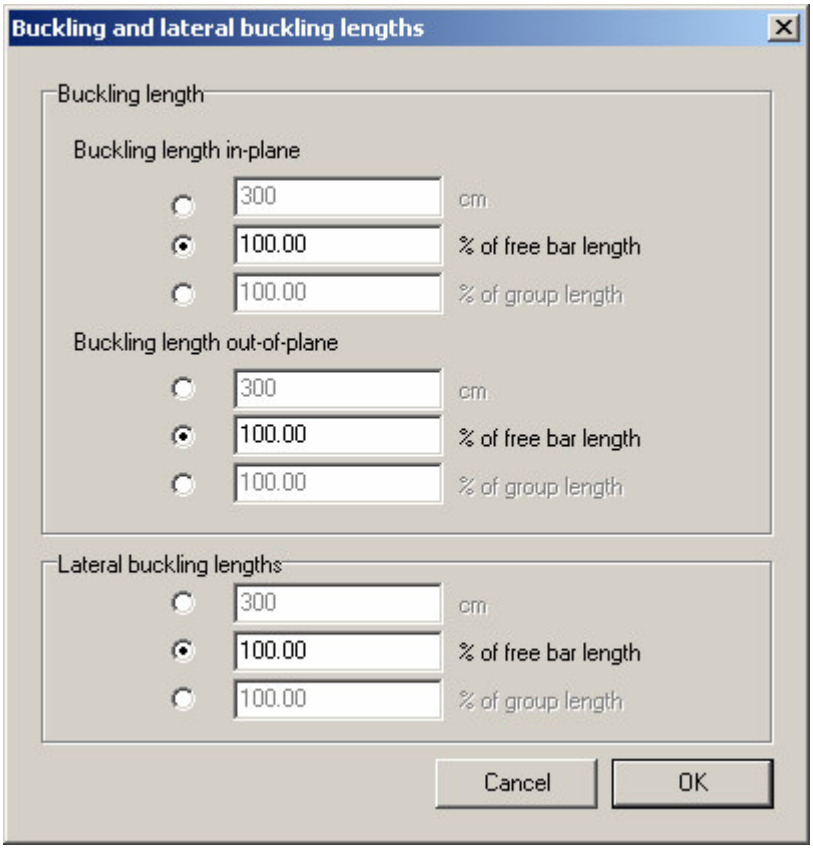
All materials in the library will then be displayed. When using steel, the grade and the production method (hot-rolled, cold-formed or welded) should also be specified.

3.1.8.3.21 Buckling and lateral buckling lengths

The buckling and lateral buckling length can easily be changed by using the icon, after having selected the appropriate member(s). Those lengths can be defined in 3 different ways:

- as an absolute value
- as a percentage of free bar length
- as a percentage of group length

As an alternative to the manual definition of buckling lengths, PowerFrame offers the possibility for automated calculation (see paragraph 3.2). Buckling lengths which have already been defined manually, will then be erased. This automated calculation is currently limited to buckling lengths and does not cover lateral buckling lengths. Those should always be defined manually.



3.1.8.4 Moving bars and nodes

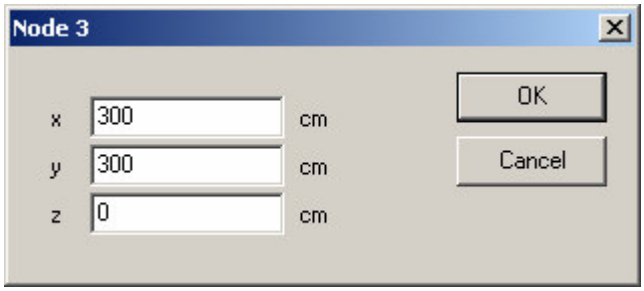
It is always possible to move bars and nodes using the translation feature incoporated in the icon toolbox of the 'Geometry' window. In addition, PowerFrame can also handle other mechanisms.

When the user has selected bars or nodes in 2D view of the model, he can move them across the window by keeping the left-hand mouse button pressed down when moving the mouse.

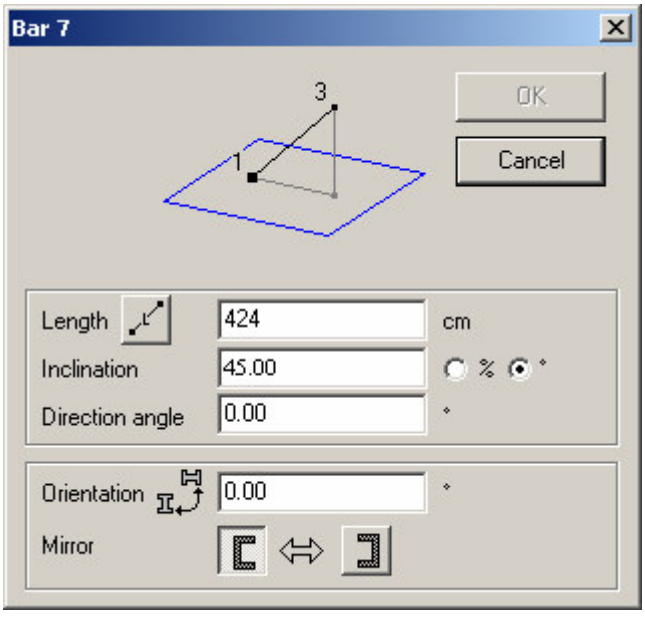
Note : this function can be activated/disactivated through the main menu function 'Edit – Preferences'.

3.1.8.5 Modifying bars and nodes

Double-click on a bar or a node to make the appropriate dialogue window appear. Double-clicking on a node gives access to an editor through which the nodal coordinates can be modified.



Similarly,, double-clicking on a bar will allow its length, its slope and the orientation of its cross-section to be changed.



Length and slope will be modified considering one end of the bar remains fixed, that end which is closest to the point of the bar on which the user has double-clicked. The other bar end will be moved according to the modifications as defined for length and/or slope.

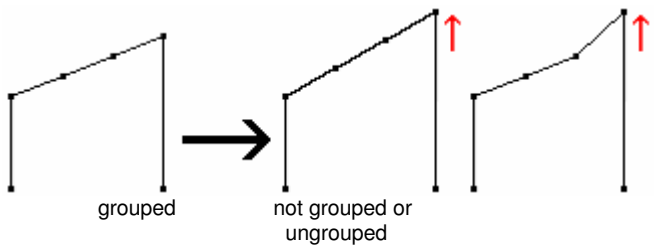
User can either specify the actual length of the member or the projection of the length on the horizontal axis, depending on the icon or (both icons relate to the same button).

3.1.8.6 Grouping and ungrouping bars

PowerFrame considers bars between any 2 nodes that are connected by a line. In case new bars have to be connected between the end nodes of an existing bar, PowerFrame will automatically create intermediate nodes and will divide the original members into individual segments. Subdividing a member into individual bars will not influence the results of the analysis (internal forces and stresses), as members are by default rigidly connected at the common nodes.

Nevertheless, it is sometimes very handy to possibly consider the original member still as a whole, rather than as a series of individual segments. In other words, considering the individual segments as grouped such that model manipulations will operate on the bar as a whole rather than on the individual segments can be quite convenient. Consider, for example, the case where a specific slope should be assigned to a bar which has previously been subdivided into several segments. If this operation needs to be done on each of the individual segments, it will require quite a long time. When, on the contrary, all segments can be dealt with simultaneously, the job will rapidly be completed. Another example relates to a steel structure for which a complete design analysis has been performed, and which needs to be exported to a steel modeling software. Such software programs usually allow to generate a bill of quantities and specify the manufacturing process, together with a bar chart giving all appropriate lengths. All segments of a member should, in this case, be obviously considered as a single entity.

For example, assume a frame, in which the beams have been subdivided to allow for the definition of purlins at intermediate positions of the entire span. If all segments of the beam remain grouped, the entire bar will move – as a whole – to a new position when one of the end nodes is moved. On the other hand, when the segments are ungrouped, only the segment connected to the node which is moved will be involved in the operation, causing a discontinuity in the slope of the beam.



When a member is subdivided into several segments, all segments remain grouped by default. To ungroup them, an explicit user action is required: select the bar(s) and click on the icon to ungroup. To group individual

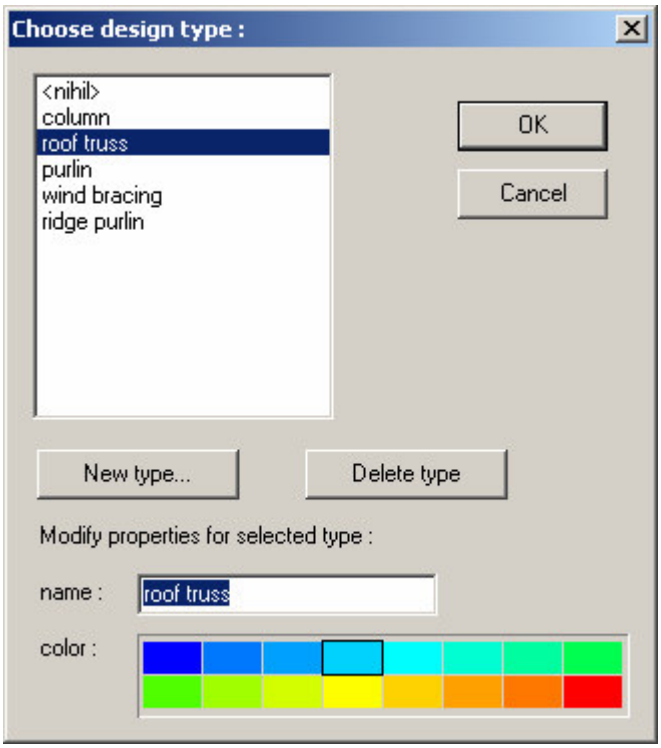
bars, click on G. Both functions are also available through the main menu ‘Screen’.

Finally, to select all members belonging to a group, select one bar of this group by keeping the ‘ALT’ key pressed down.

3.1.8.7 Element types

To further facilitate the work with PowerFrame, classes of elements can be combined into distinct TYPES. The user can create a series of different types and then assign a type to a set of selected bars. All members belonging to a specific type can easily be selected simultaneously. Just select any bar of a specific type and keep the ‘CTRL’-button pressed down. All members sharing the same type will now be selected.

Types can be defined by clicking on the T icon or through the menu entry ‘Screen – Element type…’.



A range of default types is already available, but the user can add as many types he wants. To easily recognize all elements of a specific type, a visualization color can be chosen for each type. Accordingly, all bars belonging to a specific type will no longer appear in black on the ‘Geometry’ window, but in the color previously selected.

In addition, the concept of TYPES brings a third advantage to the PowerFrame user. Upon export of a PowerFrame model to DXF, each type

will correspond to a different layer in the DXF file. Similarly, after having been imported from DXF into PowerFrame, the individual layers will be translated into different types within PowerFrame. This makes it extremely easy to select all elements of a specific type and to assign a common set of properties to all selected elements, eg cross-section properties.

3.1.8.8 Copy/paste of boundary conditions & cross-sections

If the user wants to assign the cross-section properties from one member to another or if boundary conditions corresponding to one node are also applicable to another node, the bar or node of which the user wants to copy the properties should first be selected. Keeping the right-hand side button of the mouse pressed down, makes a pull-down menu appear on the screen. The user will then select the entry “Copy cross-section” or “Copy boundary condition”, whichever is concerned.

Next, the bars or nodes to which the properties need to be transferred shall then be selected. The user will keep the right-hand mouse button pressed down and select the entry ‘Paste cross-section’ or ‘Paste boundary condition’.

3.1.9 The ‘Loads’ window

3.1.9.1 The loads icon toolbox

The first icon of the loads toolbox gives access to the definition of individual load cases (name, load coefficients, etc). The second one will allow all required load combinations to be generated. Through the third icon a dialogue window will appear which enables the definition of the correlation coefficients to be used for the calculation of design gravity loads

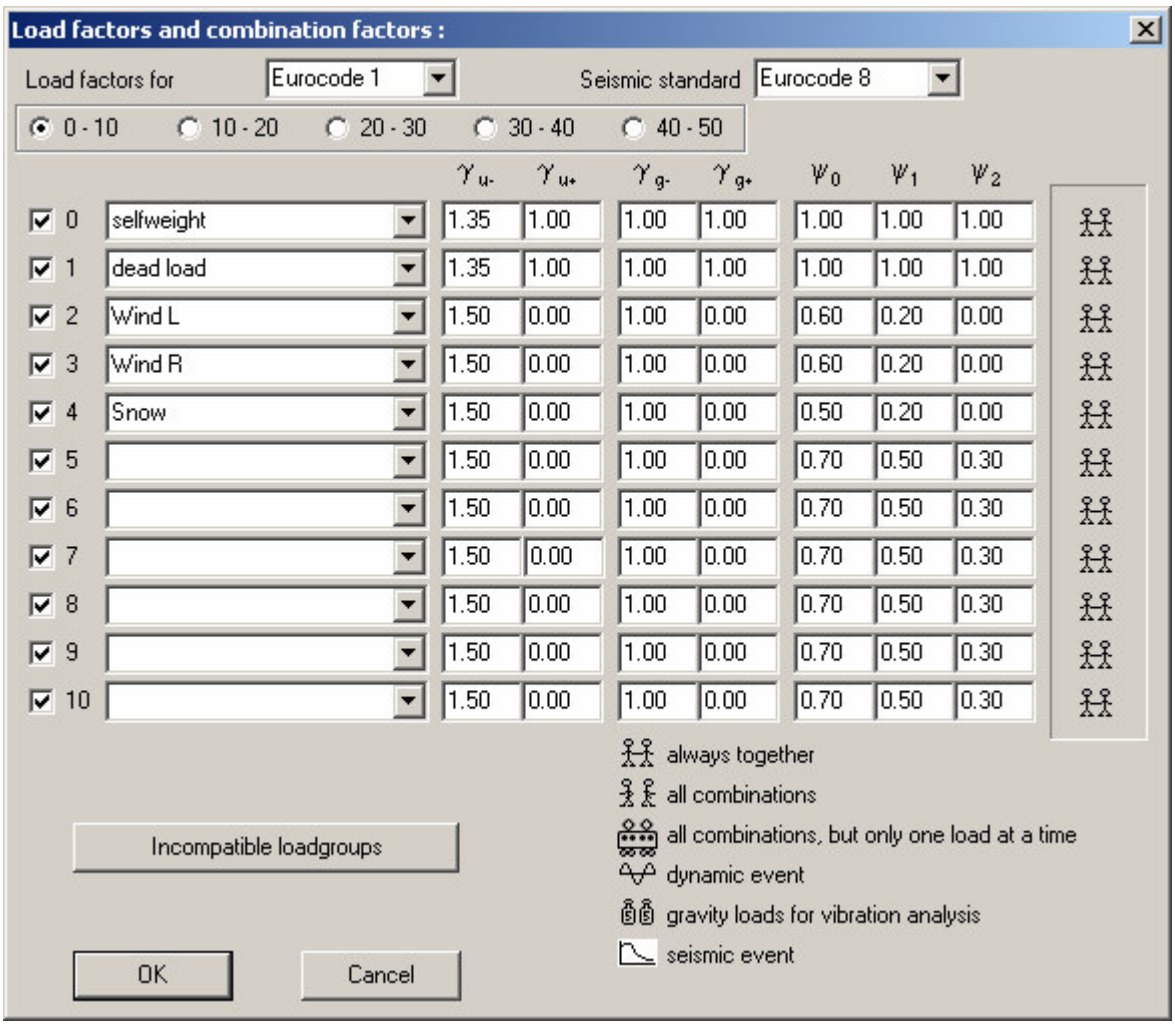
Under both icons, any of the existing load cases can be activated through a pull-down menu. Finally, under the pull-down menu, a series of icons is available to create or remove loads from the active load cases:

- the icon immediately under the pull-down menu removes all loads from the selected bars, **within the active load case.** The icon just beside allow to copy loads from one bar to another.
- the next 9 icons allow to specify different types of loads (concentrated load, moment, distributed load, displacement) at nodes or on members.
- 3 dedicated icons enable to specify a temperature increase or a pre-tensioning load on the selected bars.
- the last but one row of icons contains 2 climate load generators (wind & snow) and a capability to define distributed surface loads.
- Finally, the last icon allows to add masses to correctly model the dynamic properties of a structure.
- More explanation on all icons is to be found in the following paragraphs.

3.1.9.2 Load cases

To better understand how PowerFrame handles the load cases, the user should click on The dialogue window that now appears (see below) requires some explanation.

The pull-out menu at the top of the dialogue window allows the selection of a design code allowing the individual load cases to be specified. For each Design Code, PowerFrame will already display a couple of basic load cases. It is possible to complete this list, up to 50 different load cases. The complexity and cost of the related calculations will obviously increase with the number of specified load cases.


Load factors and combination factors :
Load factors for
Eurocode 1
Seismic standard
Eurocode 8
0 - 10
10 - 20
20 - 30
30 - 40
40 - 50
γu-
γu+
γg-
γg+
Ψ0
Ψ1
Ψ2
0 selfweight 1.35 1.00 1.00 1.00 1.00 1.00 1.00
1 dead load 1.35 1.00 1.00 1.00 1.00 1.00 1.00
2 Wind L 1.50 0.00 1.00 0.00 0.60 0.20 0.00
3 Wind R 1.50 0.00 1.00 0.00 0.60 0.20 0.00
4 Snow 1.50 0.00 1.00 0.00 0.50 0.20 0.00
5 1.50 0.00 1.00 0.00 0.70 0.50 0.30
6 1.50 0.00 1.00 0.00 0.70 0.50 0.30
7 1.50 0.00 1.00 0.00 0.70 0.50 0.30
8 1.50 0.00 1.00 0.00 0.70 0.50 0.30
9 1.50 0.00 1.00 0.00 0.70 0.50 0.30
10 1.50 0.00 1.00 0.00 0.70 0.50 0.30
always together
all combinations
all combinations, but only one load at a time
dynamic event
gravity loads for vibration analysis
seismic event
Incompatible loadgroups
OK
Cancel

The line just below the pull-out menu gives access to any of the 50 possible load cases. For practical reasons, only 10 load cases are displayed at a time. The radio buttons should be used to switch to another group of 10 load cases.

Each line corresponds to an individual load case. In the first column of each line, the user can select or unselect the corresponding load case. Unselecting a specific load case will not affect the actual load definitions within that case, but will eliminate the load case from the load combinations that are created. Also, loads that are part of a load case which is not selected will be displayed in gray on the 'Loads' window, as opposed to purple for the selected load cases.

Use the pull-downs of the second column to choose the type of load for a specific load case (dead load, superimposed dead load, live load, traffic load, wind, snow, …). It can be noted that the safety factors and combination coefficients in the subsequent columns will automatically change in accordance with the selected type and Design Standards. It should also be noted that the name of the load case can be changed any time by directly editing the related field.

Some more information will be given now about safety factors and combination coefficients. Whether a load is permanent or variable, it can have either a favorable or unfavorable effect on a specific design response (deflection, bending moment, shear force,…) at a selected location of the analysis model, depending on where those loads are actually located. Therefore, Design Standards prescribe distinct safety factors for favorable and unfavorable impact on design response. In general, the Eurocode specifies a safety factor of 1.35 or 1.00 for permanent loads respectively in case of an unfavorable impact on design response or a favorable impact on design response. For live loads, these coefficients become 1.50 and 0.00. Other National Standards and Codes may specify slightly different coefficients.

Both columns γ_u correspond to ultimate limit states (ULS), while both columns γ_g correspond to serviceability limit states (SLS). Within both types, the index "+" relates to a favorable impact of the load while the index "-" relates to an unfavorable impact.

The next 3 columns contain combination coefficients:

- ψ_0 is the combination coefficient applied to a specific load case for the fundamental combinations in ultimate limit states and for those rare combinations in serviceability limit states for which the related load case has the most unfavorable impact on design response;

- ψ_1 is the combination coefficient applied to a specific load case for accidental combinations at ultimate limit states and for those frequent combinations in serviceability limit states for which the related load case has the most unfavorable impact on design response;

- ψ_2 is the combination coefficient applied to a specific load case for the quasi-permanent combinations in serviceability limit state. For the accidental combinations in ultimate limit states and for frequent combinations in serviceability limit states, this coefficient is applied when another load case has a more unfavorable impact on design response.

Finally, the column at the right-hand side contains an icon which changes when selected. This icon helps to define the simultaneous loads on multiple spans. Situation can occur in which the live load acting on a continuous beam with 3 spans, is not applied at the same time on all spans. When considering, for instance, the bending moment in the central span, the most unfavorable condition is met when the live load is not applied on that particular span. In

the table below, are indicated systematically which combinations of loads are considered for different simultaneity conditions – again for the example of a 3-span beam. Application of the unfavorable safety coefficient (γ_-) is indicated by a "✓", while application of the favorable coefficient (γ_+) is indicated by a "-".

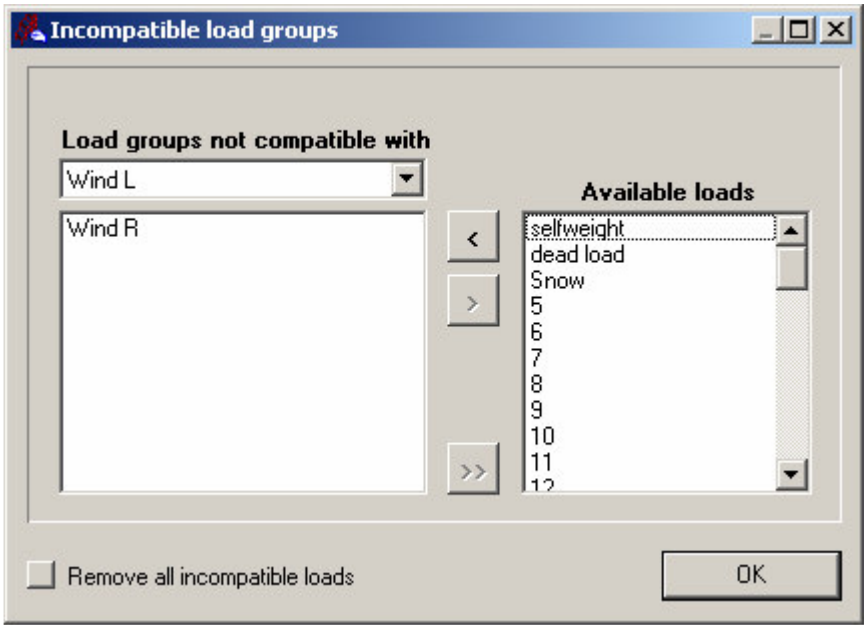
Simultaneity of loads		Number of situations		1st span	2nd span	3rd span
	always together	2	1	-	-	-
			2	✓	✓	✓
	all combinations	$2^3 = 8$	1	-	-	-
			2	✓	-	-
			3	-	✓	-
			4	-	-	✓
			5	✓	✓	-
			6	✓	-	✓
			7	-	✓	✓
			8	✓	✓	✓
	all combinations, but only 1 load at a time	3 + 1 = 4	1	-	-	-
			2	✓	-	-
			3	-	✓	-
			4	-	-	✓

The number of combinations increases rapidly in function of the number of spans to be considered. For an 8 spans continuous beam where each one can be loaded or unloaded independently, the number of possible load combinations amounts to $2^8 = 256$! When a non-linear analysis is performed, the calculation of this specific combination of loads already requires 256 different analyses. In case of a linear analysis in which the principles of linear superposition are valid, 8 analyses only are enough. To limit RAM requirements and calculation time, PowerFrame does not allow for a non-linear analysis when more than 8 members are loaded within a specific load case in which all possible combinations ( ) should be considered. An analysis is non-linear when at least one of the following conditions is fulfilled:

- the analysis model includes supports which can only provide tensile OR compressive reaction forces;
- the analysis model includes tie rods;
- a 2nd order analysis strategy has been selected;
- the analysis strategy considers the effect of structural imperfections.

Only when the 4 preceeding conditions are not met simultaneously, a load case can be defined in which more than 8 bars are loaded and for which the user specifies that all possible combinations ( ) should be considered.

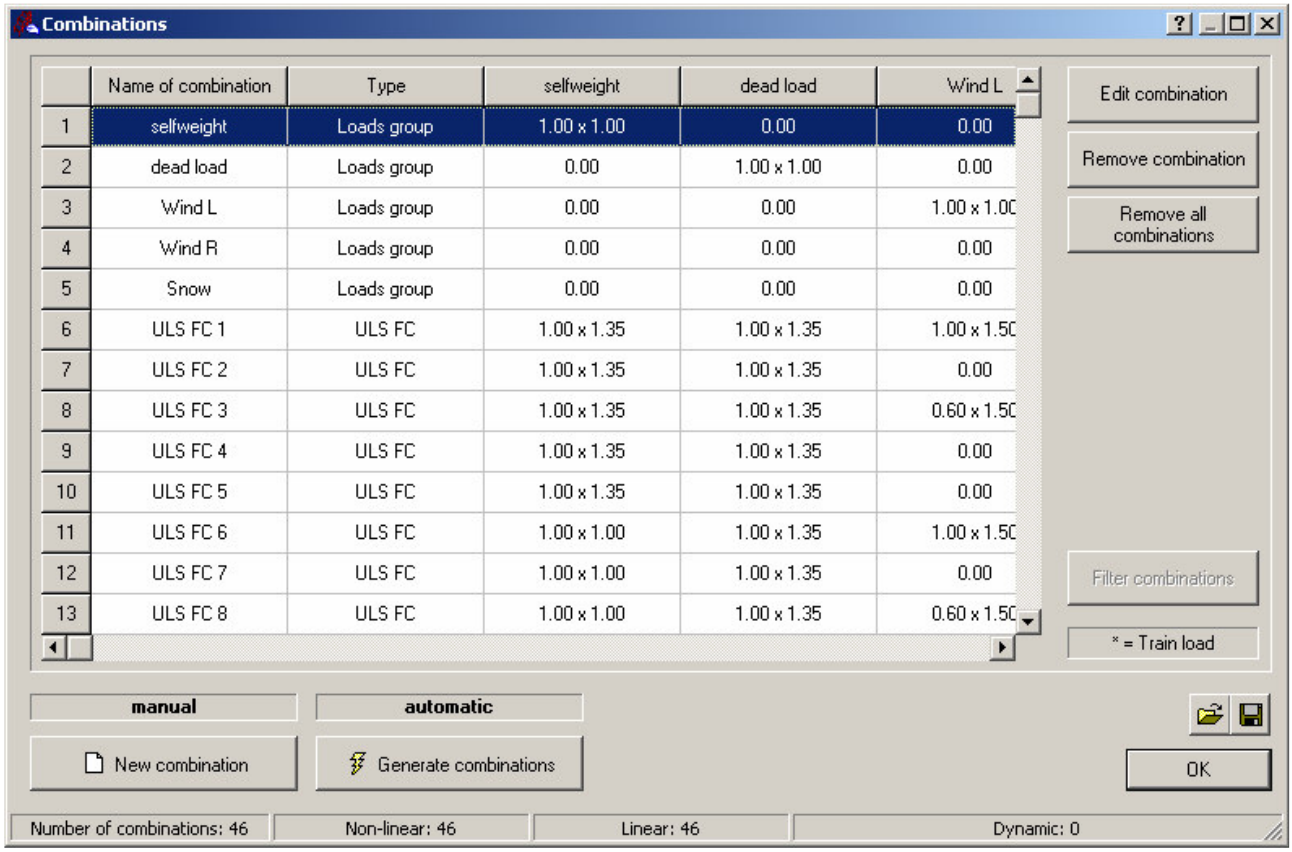
One final point should be brought to the attention of the user, concerning more particularly the topic of incompatible load cases. To make specific load cases incompatible by specifying they can never be present together in any load combination, the button 'Incompatible load groups' should be used, which opens the dialogue window shown below, which will now be further explained



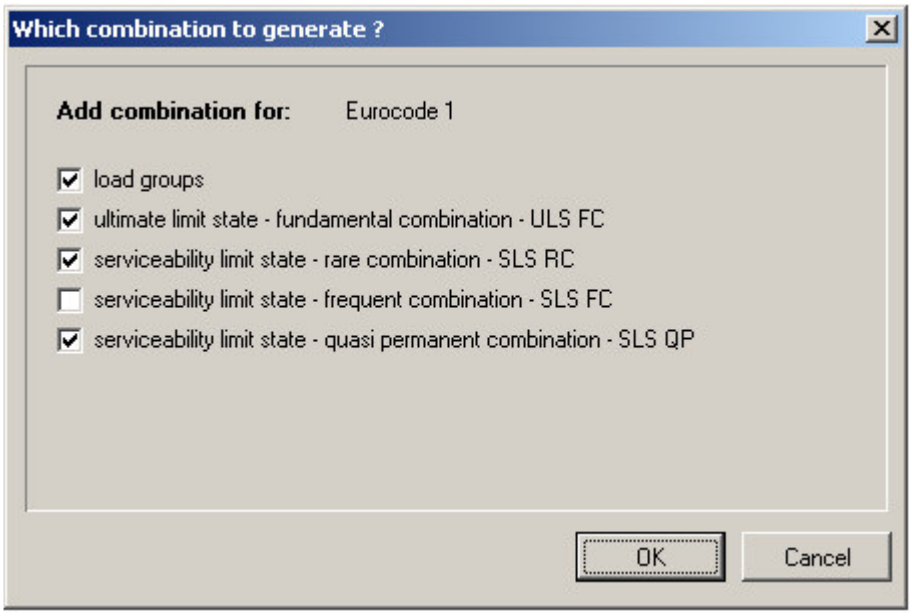
To make a particular load case incompatible with a range of other cases, select the name of the load case using the pull-down. The column below this pull-down displays the load cases which have already been declared incompatible with the selected one. To add a new incompatible load case to the list, select it from the right-hand column and move it to the left-hand side using the left arrow. Similarly, any load case can be removed from the list of incompatible load cases by selecting it in the left-hand column and moving it back to the right-hand side using the appropriate arrow in the dialogue. To remove all incompatible load cases from the list in one single operation, just select the field "Remove all incompatible loads" at the bottom.

3.1.9.3 Load combinations

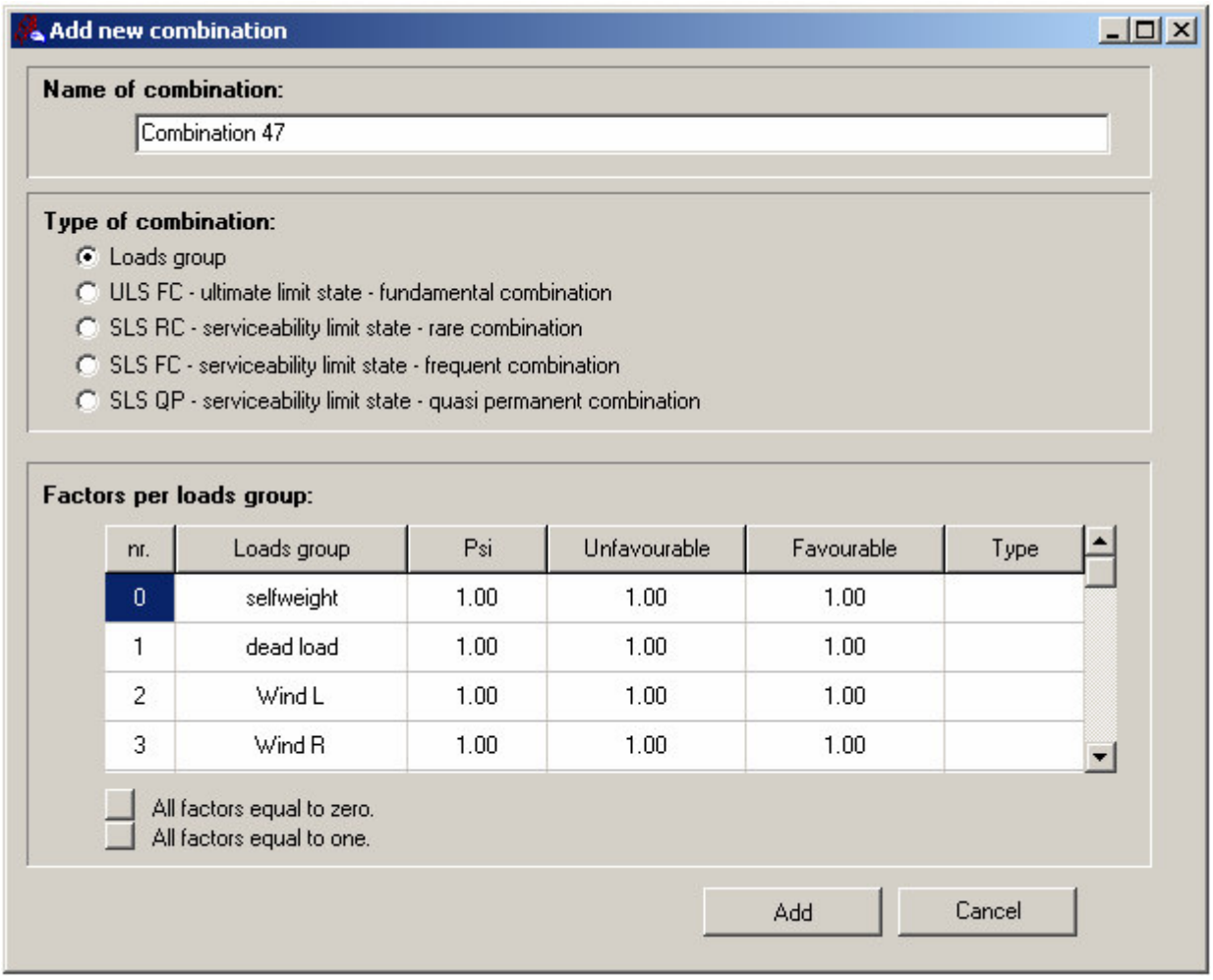
Once loads and load cases have been completely defined, PowerFrame allows an automatic or manual specification of load combinations, using all previously defined factors. To start this process and access the dialogue box shown below, the icon in the toolbox should be selected.


Combinations
Name of combination
Type
selfweight
dead load
Wind L
1 selfweight Loads group 1.00 x 1.00 0.00 0.00
2 dead load Loads group 0.00 1.00 x 1.00 0.00
3 Wind L Loads group 0.00 0.00 1.00 x 1.00
4 Wind R Loads group 0.00 0.00 0.00
5 Snow Loads group 0.00 0.00 0.00
6 ULS FC 1 ULS FC 1.00 x 1.35 1.00 x 1.35 1.00 x 1.50
7 ULS FC 2 ULS FC 1.00 x 1.35 1.00 x 1.35 0.00
8 ULS FC 3 ULS FC 1.00 x 1.35 1.00 x 1.35 0.60 x 1.50
9 ULS FC 4 ULS FC 1.00 x 1.35 1.00 x 1.35 0.00
10 ULS FC 5 ULS FC 1.00 x 1.35 1.00 x 1.35 0.00
11 ULS FC 6 ULS FC 1.00 x 1.00 1.00 x 1.35 1.00 x 1.50
12 ULS FC 7 ULS FC 1.00 x 1.00 1.00 x 1.35 0.00
13 ULS FC 8 ULS FC 1.00 x 1.00 1.00 x 1.35 0.60 x 1.50
Edit combination
Remove combination
Remove all combinations
Filter combinations
* = Train load
manual
automatic
New combination
Generate combinations
OK
Number of combinations: 46
Non-linear: 46
Linear: 46
Dynamic: 0

Initially, the table in the above dialogue table does not contain load combinations. When the user clicks on the 'Generate combinations' button, PowerFrame will request to specify the combinations to be created for the actual design calculations later on: individual load cases, ultime limit state (ULS) combinations and serviceability limit state (SLS) combinations (SLS QP) for quasi-permanent combinations, (SLS RC) for rare combinations and (SLS FC) for frequent combinations.


Which combination to generate ?
Add combination for: Eurocode 1
load groups
ultimate limit state - fundamental combination - ULS FC
serviceability limit state - rare combination - SLS RC
serviceability limit state - frequent combination - SLS FC
serviceability limit state - quasi permanent combination - SLS QP
OK
Cancel

If the user wants to define a particular load combination manually, the 'New combination' button will be used. PowerFrame will then ask to specify all coefficients manually, rather than taking them from a pre-defined list.


Add new combination
Name of combination:
Combination 47
Type of combination:
Loads group
ULS FC - ultimate limit state - fundamental combination
SLS RC - serviceability limit state - rare combination
SLS FC - serviceability limit state - frequent combination
SLS QP - serviceability limit state - quasi permanent combination
Factors per loads group:
nr. | Loads group | Psi | Unfavourable | Favourable | Type
0 | selfweight | 1.00 | 1.00 | 1.00
1 | dead load | 1.00 | 1.00 | 1.00
2 | Wind L | 1.00 | 1.00 | 1.00
3 | Wind R | 1.00 | 1.00 | 1.00
All factors equal to zero.
All factors equal to one.
Add
Cancel

Specify the name of the combination to be created, the type of combination and the coefficients related to the individual load cases.

The user will then notice that the table shown above includes all load cases that have previously been defined and selected. For each load case, 3 columns have to defined: combination coefficient psi (ψ), unfavourable (γ_-) and favourable (γ_+) safety factor.

To remove a combination from the list of existing ones, select it in the table and then use the button 'Remove combination'. If, on the other hand, the user wishes to modify an existing combination, he should use the 'Edit combination' button. Choosing 'Remove all combinations' will then delete all combinations from the table. Always remember that creating new combinations will ADD them to the list of existing ones. Accordingly, if the user wants to replace an existing list of combinations by a new series of combinations, he should first always remove all existing combinations.

Finally, if the user wants to save the definition of load combinations to an external file, he can do so by using the icon. He can always retrieve this definition from the external file using the icon.

3.1.9.4 Defining loads

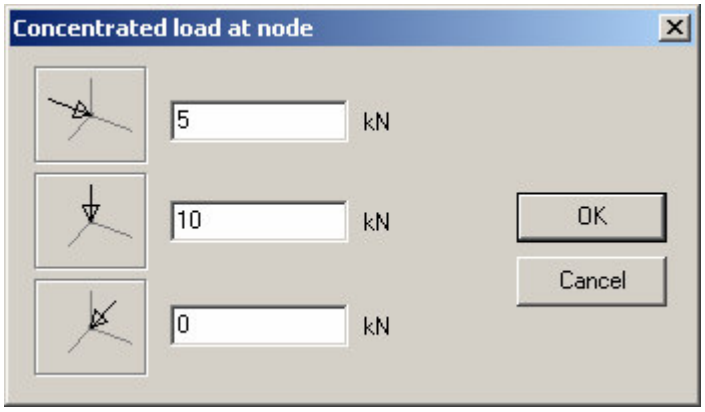
Before discussing the different types of individual loads in detail, let us shortly review how loads are removed or modified. To remove loads within the active load case, select one or more members or nodes where loads are applied and click on the icon. If on the other hand, the user wishes to modify the value of loads applied on a given member or node, he should double-click on this very bar or node. A table will appear presenting all selected loads. Values can be changed directly in this table.

3.1.9.4.1 Loads at nodes

PowerFrame allows to apply the following load types at the nodes of a model: concentrated load, moment load and imposed displacement.

Concentrated loads

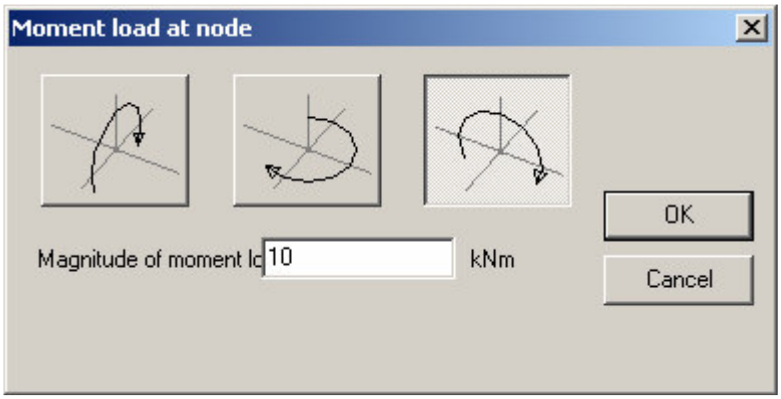
Having selected the implied node(s), click on within the icon toolbox to introduce a concentrated load at the selected node(s).



The graphs in the above dialogue will adapt automatically to the viewpoint in the 'Loads' window. Each line corresponds to one of the global coordinate axes. If the user work in a 2D view, only the lines that correspond to the working plane will be active.

Moments

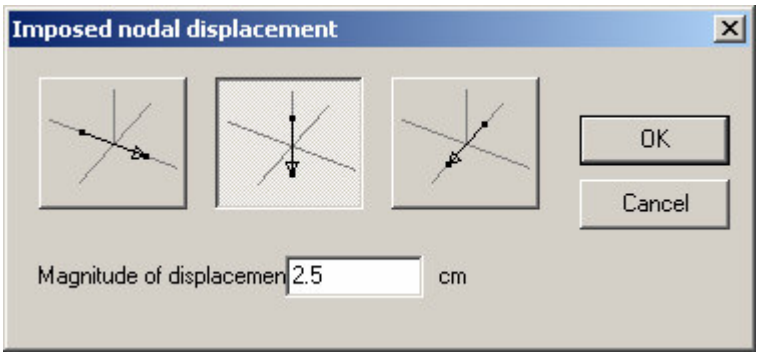
Having selected the implied node(s), click on to define the couple to be applied.



Again, the graphs in the dialogue table will adapt automatically to the viewpoint in the 'Loads' window.

Imposed displacements

Having selected the implied node(s), click on to define the displacement to be imposed at these selected node(s).



Again, the graphs in the dialogue table will adapt automatically to the viewpoint in the 'Loads' window.

Note : To impose a displacement at a node in a given direction, the user should have constrained the corresponding DOF in the 'Geometry' window.

3.1.9.4.2 Loads on members

Loads on members can be defined in 2 different ways:

- either in accordance with the local coordinate axes associated to the member

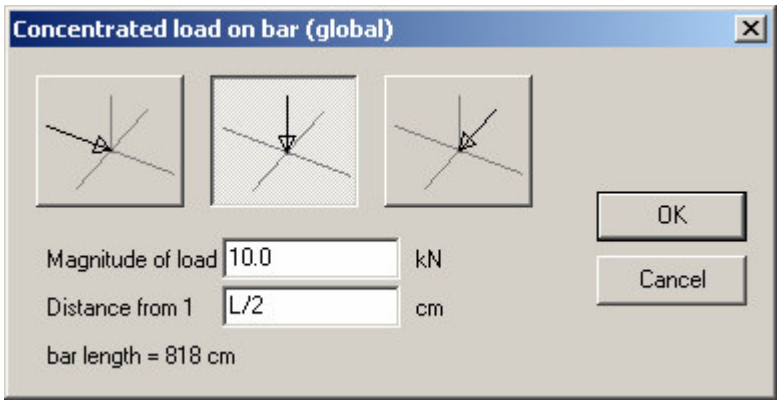
- or in accordance with the global coordinate axes of the structural model.

In addition, the following load types can be assigned to members :

- concentrated load at an intermediate point
- moment at an intermediate point
- distributed load (either uniform or trapezoidal).

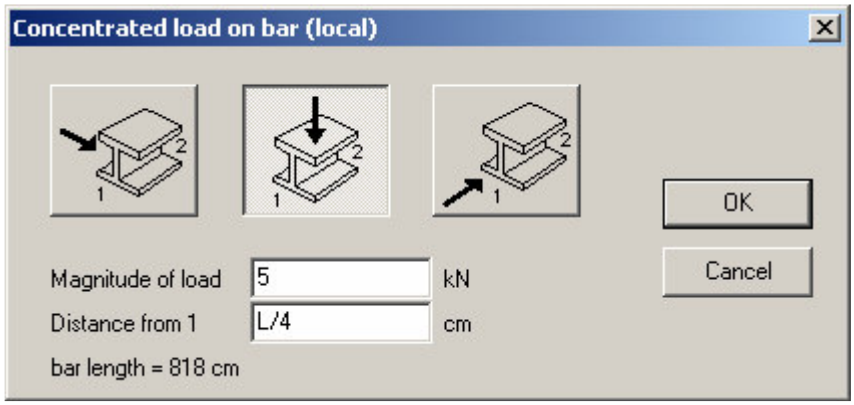
Concentrated loads

To apply a concentrated load at an intermediate point of a bar according to the global coordinate system axes, select the bar and click on .



The icons will adapt automatically to the viewpoint in the 'Loads' window to make a correct and easy definition, minimizing error risks. The field in which the relative position of the load can be introduced along the bar axis (using the end node with smallest x-value as a reference) accepts values that are defined as a fraction of the bar length L.

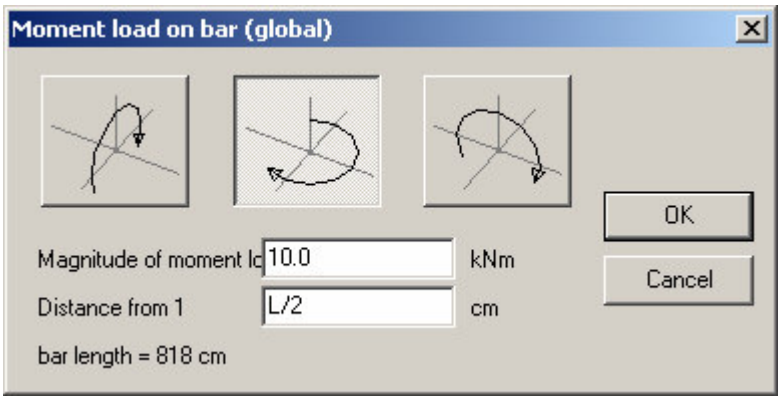
To define a concentrated load in the local coordinate system of a bar, select the bar and then click on .



The definition process is further completed as previously described for concentrated loads defined in the global coordinate system.

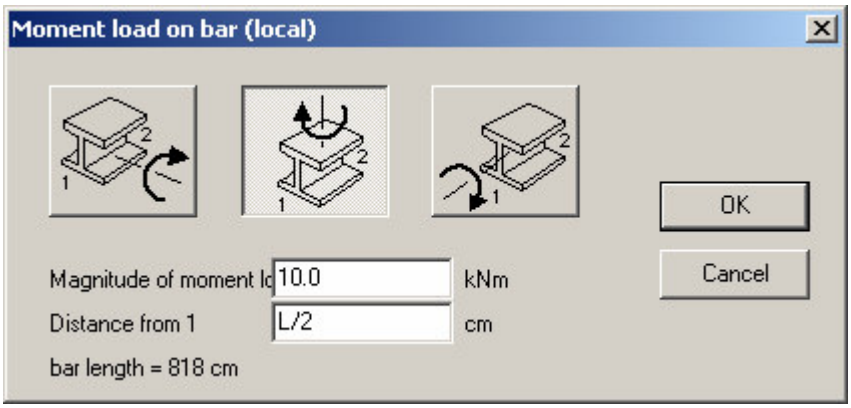
Moment

To specify a concentrated moment on one or more selected bars along the global coordinate system axes, click on .



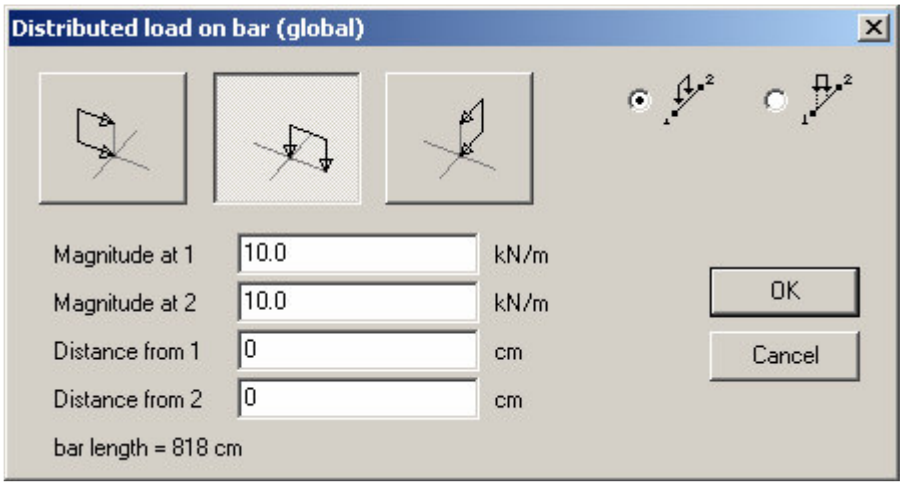
The icons will adapt automatically to the viewpoint in the 'Loads' window to make a correct definition as easy as possible, minimizing error risks. The field in which the user can introduce the relative position of the moment along the bar axis (using the end node with smallest x-value as a reference) accepts values that are defined as a fraction of the bar length L.

To define a concentrated moment on one or more selected bars along the local axes of the bar, click on . To further edit the dialogue, the same principle can be used as for the definition of moments in the global coordinate system.



Distributed loads

To define a distributed load on selected bars (or parts thereof) in the global coordinate system of the bar element, click on .

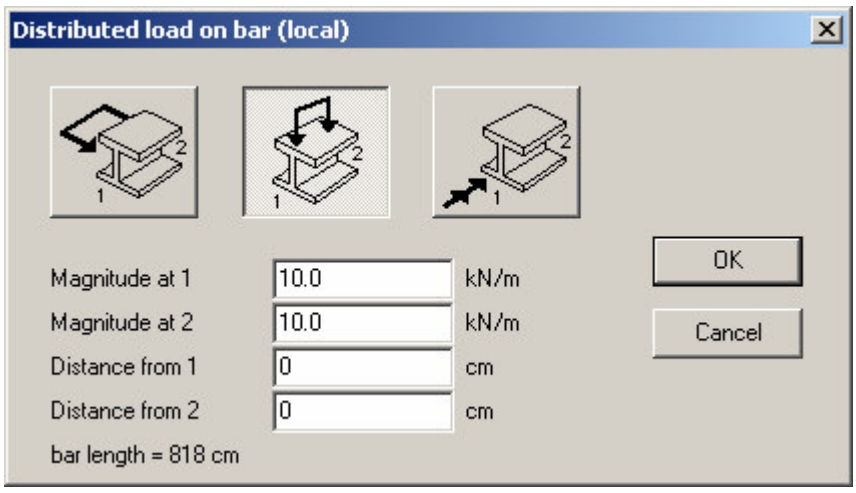


The first 2 editor fields allow to introduce the end values of the distributed load. If only the first field is used, then the second field will automatically be equal to the value entered in the first field, thus defining a uniform load on the bar, or part thereof. If the user explicitly enters a different value in the second field, a trapezoidal load is applied on the bar.

Similarly, 2 editor fields allow the user to enter the distance of the load application points along the axis of the bar. Those distances are defined relative to the end points 1 and 2 of the bar.

If the user wants to define the load per unit distance along the horizontal projection of the bar, the icon is to be selected in the right-hand upper corner of the dialogue box. In this case, the load per unit length along the bar axis will decrease as the slope of the bar increases. If the user selects the option , load values are considered to be specified per unit length along the bar axis.

To define a distributed load on selected bars, or part thereof, according to the local bar axes, click on .

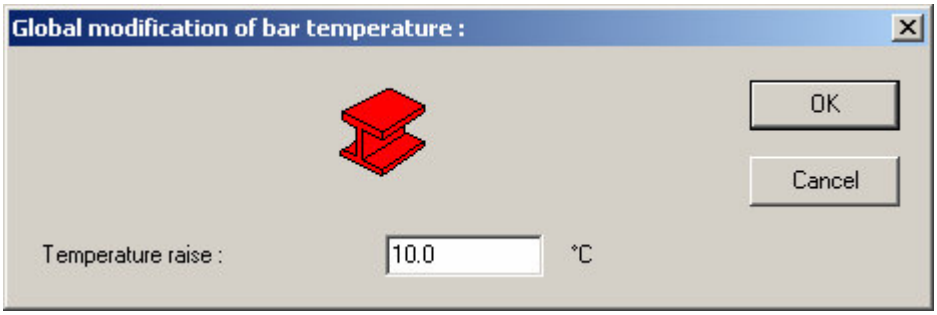


To further edit the dialogue, the same principles can be used as for the definition of distributed loads in the global coordinate system, except for the possibility to define load values per unit length of horizontal projection as this does not make sense in such a situation.

3.1.9.4.3 Temperature exposure

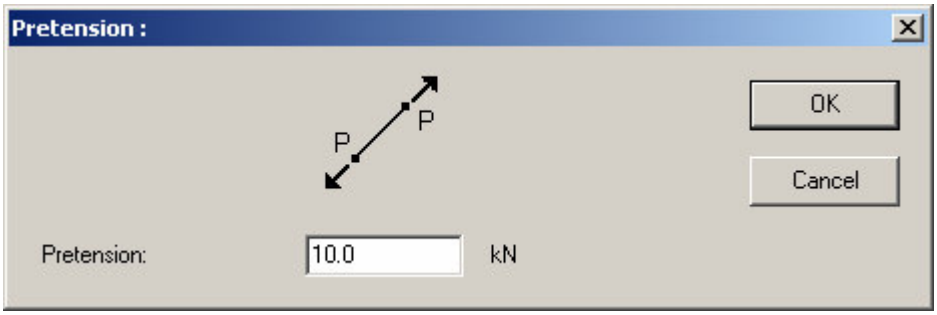
PowerFrame allows to handle a global temperature exposure on selected bar(s). The corresponding mechanical load is calculated using the thermal dilatation coefficient of the material used for the selected bar(s), considering the stresses that are induced in the bar due to the restrained dilatation following a temperature variation. This coefficient is stored with other material properties in the material database, and can then be modified whenever required.

To define the temperature value on selected bar(s), click on the icon and PowerFrame will ask to specify the temperature raise.



3.1.9.4.4 Pretensioning load

In structural modeling applications, situations may arise in which bars or cables are subject to an initial pre-tensioning (for example, a cable which is to remain under tension during operating conditions). To define the pre-tensioning level applied to the selected bars, click on .



IMPORTANT NOTE : pretensioning loads are to be applied only at nodes where the structure can develop a reaction force. At free nodes, this is impossible. Hence, the application of a pretensioning load on a bar with a free node will have no impact on the structural behaviour.

3.1.9.4.5 Generating climate loads

PowerFrame is enriched with an automatic entry for wind and snow loads. Both cases require the selection of a complete set of bars that comply with the following requirements:

- all selected bars are within one single vertical plane
- all selected bars should represent a closed structure with respect to ground level.

The aim of the climate load generators is to introduce values as close as possible to the requirements of the various Codes and Standards.

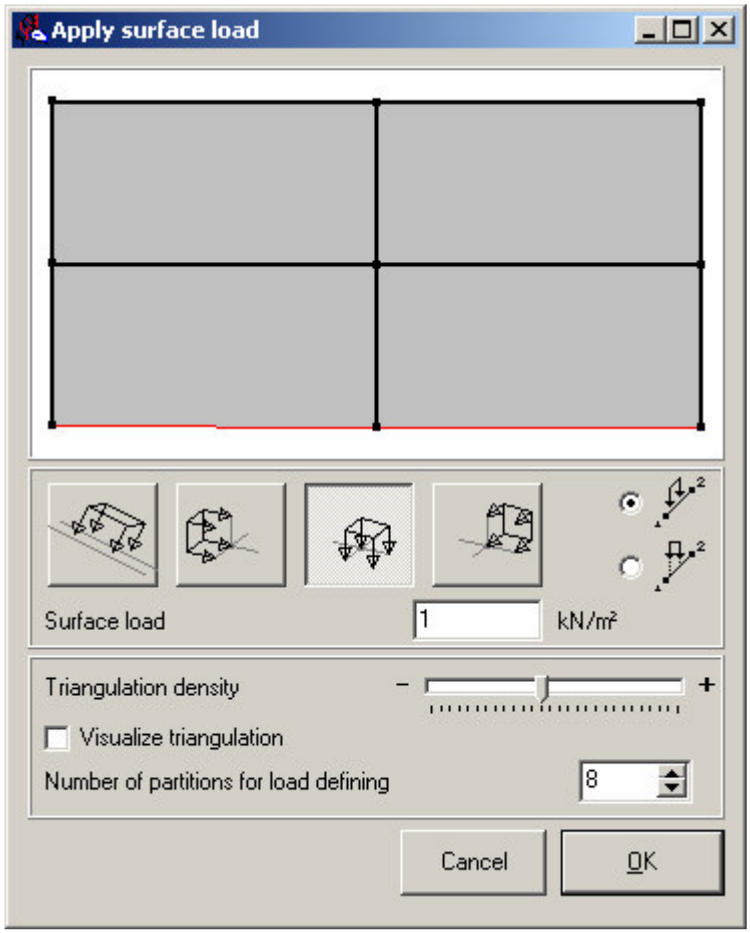
Having made a valid selection of bars, click on to launch the wind generator or on to launch the snow generator. If the selection that has been made is invalid, the icons will remain grayed out.

Once the appropriate generator has been launched, a window will ask to specify a design Standard. Next, a dedicated dialogue field will prompt the user to define all parameters related to the climate load.

For more details, please refer to the section of the PowerFrame manual dealing with this subject.

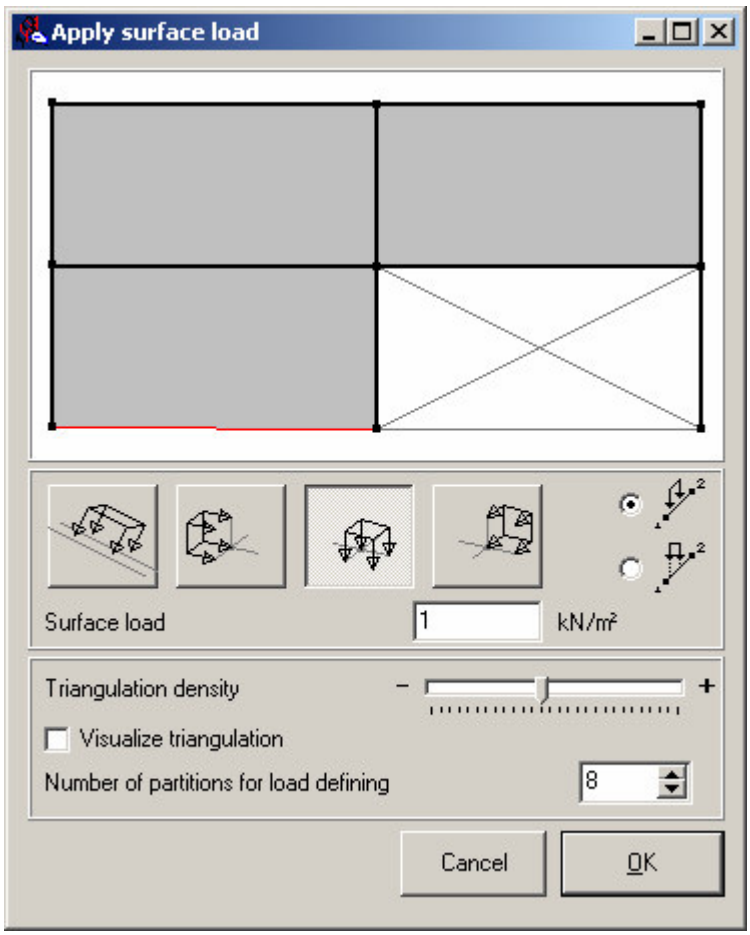
3.1.9.4.6 Surface loads

Use the icon to specify distributed loads on a surface defined by the selected bars, and to automatically transfer those surface loads to distributed line loads on bar elements.



In the upper half of this dialogue window, a view is shown of the surface (in gray) defined by the selected bar elements. All selected bars are drawn in black. In case those bars do not define a closed surface, PowerFrame will automatically add red-colored borders so as to create a closed surface by itself. If those borders do not meet your requirements, just deselect them through a simple mouse click. Another border will then automatically be proposed, which can again be accepted or rejected.

A simple mouse click on any of the black bars allows you to guard such a bar from carrying any load. Such a bar is colored gray in the above visualization, and it will be not be considered by PowerFrame during the transformation of the surface load to equivalent line loads.

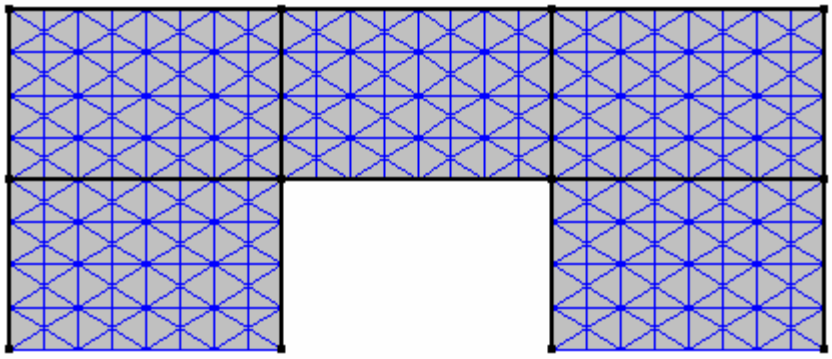


In the next field, the user defines the direction and value of the surface load. Just as for any other type of load previously discussed, the load icons are adapted automatically to the viewpoint of the ‘Loads’ window.

It should be noted that with surface loads, the definition of load per unit surface is possible either along the surface itself or along its horizontal projection.

Let’s now have a closer look how the surface loads are distributed towards the selected bar elements.

First of all, a triangular surface mesh is created for which mesh size depends on the triangulation density specified by the user. The triangulation created by this automatic meshing procedure can be visualized by activating the option ‘Visualize triangulation’.



Next, PowerFrame will calculate for each mesh triangle the resultant P of the surface load acting on that triangle. It will then investigate the influence of such a resultant force 'P' on all selected lines which belong to the closed surface.

Assuming that

- ✓ a given line is divided in several nodes 1 until k,
- ✓ the distance between each of these nodes (node number j, for instance) and the centre of gravity of the active triangle is described as d_i,

then the load P is now distributed to the lines according to:

$$F_i = P \times \frac{1/d_i}{\sum_j 1/d_j}$$

Applying this rule for all triangles and all mesh nodes, the surface load will be converted into distributed line loads on all selected bars.

3.1.9.4.7 Modifying or removing loads

To remove loads within the active load case, the user selects one or more bars/nodes where loads are applied and clicks on the icon. If on the other hand, he wishes to modify the value of loads that are applied on a given bar or node, double-click on the bar or node. A table will appear displaying all selected loads. Values can be changed directly in this table.

3.1.9.4.8 Copy/paste of loads

If the user wants to copy loads applied at one bar to another bar and/or another load case, he can use the functions 'Copy loads' & 'Paste loads'. The user is first to select the bar from which he wants to copy loads, then use the

right-hand mouse button to make a menu appear on the screen, allowing to use the ‘Copy loads’ command.

Now the user selects the load case to which he wants to paste the loads (if different from the active load case) and then selects the bars to which he wants to actually paste those loads. He should use again the right-hand mouse button to make the same menu appear on the screen, but now selects the ‘Paste loads’ command.

3.1.9.4.9 Dynamic mass

PowerFrame enables a modal analysis to be performed on the structural model and to calculate its fundamental natural frequency, including the impact of lumped masses that are added to correctly model the dynamic properties of a structure. To define these masses, select the nodes where the masses must be added and then click on the icon.

Note : This icon is only active if the load case ‘Gravity loads for vibration analysis’ is activated through the pull-down menu.

3.1.10 The ‘Plot’ window

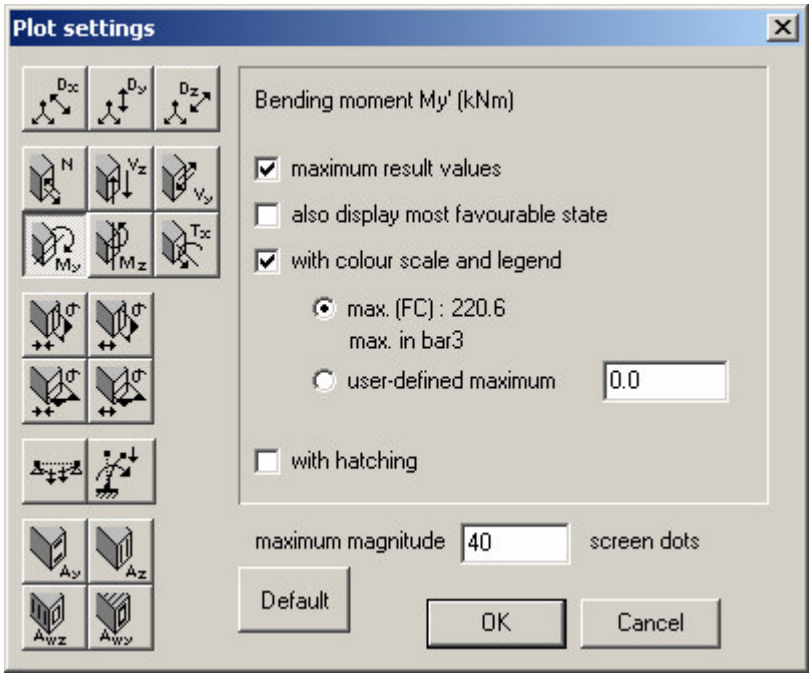
The ‘Plot’ window allows to visualize graphically all analysis results. If no analysis has been performed yet, or if changes have been made to the analysis model without re-running the analysis, this window will be empty.

In case analysis results are available in the ‘Plot’ window, the user first chooses the load case or load combination using the pull-down menu in the icon toolbox.

Note that with any type of results shown by PowerFrame, a color scale is always associated at the right-hand side of the ‘Plot’ window. The range of this color scale is automatically adapted to the result values referring to the visible bars.

3.1.10.1 Plot parameters

The plot parameters for the different types of results can be modified in a dedicated dialogue window which is accessed through the main menu ‘Show – Plot parameters…’ or by selecting the icon in the upper icon toolbar.



The dialogue window which appears contains the majority of the icons that are also present in the icon toolbox of the 'Plot' window.

For each icon in the dialogue box, the user can specify the plot parameters separately, by clicking on the corresponding icon to access the parameters and edit the fields at the right-hand side. .

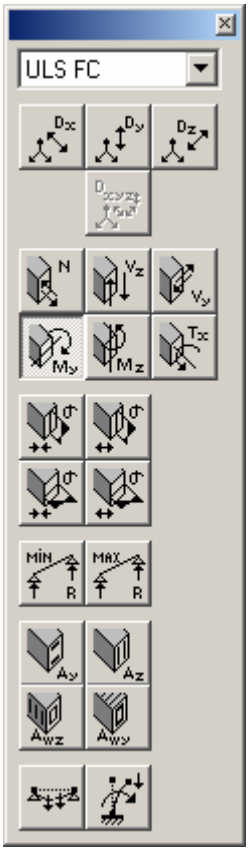
If the first option has been selected, maximum values will appear on the screen next to the results curve, for all visible bars. The second option allows to show or hide the curve giving the minimum result values.

If the third option is selected, results will be presented on a colored scale having a range which depends either on the maximum result values or on pre-defined result values. If this option is not selected, results will be shown using a monochrome display mode while the color scale legend will disappear from the 'Plot' window.

The last option allows the user to specify whether he wants to show the surface between the results curve and the undeformed bar with or without hatching.

Finally, the editor field located at the bottom of the dialogue window is used to define the number of screen dots to represent the maximum results with respect to the undeformed bar.

3.1.10.2 The plot icon toolbox



Moving over the icon toolbox downwards, the following entities are encountered :

Pull-down menu

The pull-down menu contains the complete set of combinations that was generated in the 'Loads' window, along with the appropriate envelopes.

The envelopes are named 'ULS' for the ultimate limit states, 'SLS RC' for the serviceability limit states, rare combinations, and 'SLS QP' for the serviceability limit states, quasi-permanent combinations. Those envelope curves are of course only available if at least one load combination is present in the corresponding group.

Deflections

The first 4 icons below the pull-down allow to visualize the deflections of any structural bar.

Using the icon will plot only the X-component of bar deformations – in the global coordinate system of the model. The icons and perform the same operation, but now with respect to global Y- and Z-components of deformations.

Below those 3 icons, allows to plot the complete deformation in the global coordinate system. This icon is not active for envelope curves, as a single line does not allow all possible deformations of the system to be shown.

Internal forces

Using , plots the axial forces in all bars, both tensile and compressive forces at the same time (negative values correspond to compressive forces).

The shear force along the strong axis of the bar cross-section is displayed by using icon, while the icon plots the shear force along the weak axis.

Similarly, bending moments with respect to strong & weak axis of the bar cross-section are plotted using and . Bending moments are shown at the fiber of the bar which is subject to tension.

The last icon in the series related to internal forces, corresponds to the torsional moment : .

Elastic stresses

Elastic stresses are not available for reinforced concrete bars.

: allows to plot the maximum compressive stress due to an axial force N and a bending moment M with respect to the strong axis of the bar cross-section.

: allows to plot the maximum tensile stress due to an axial force N and a bending moment M with respect to the strong axis of the bar cross-section.

: allows to plot the maximum compressive stress due to an axial force N and a bending moment M with respect to the weak axis of the bar cross-section.

: allows to plot the maximum tensile stress due to an axial force N and a bending moment M with respect to the weak axis of the bar cross-section.

Reaction forces

If one series of reaction forces only is available, the icon will allow to plot reaction forces at all supports. However, if a reaction force at a support has a minimum and a maximum value, two icons are visible : and .

Concrete reinforcement

In case the PowerFrame analysis contains concrete cross-sections and if reinforcement has already been calculated with respect to a selected Standard, the reinforcement icons will become active. A total of 4 icons will be available :

: to plot longitudinal reinforcement quantities parallel to the strong axis of the cross-section, at the top and bottom fibers of a member;

: to plot longitudinal reinforcement quantities parallel to the weak axis of the cross-section, at the front and rear fibers of a member;

: to plot transverse reinforcement quantities parallel to the weak axis of the cross-section. This reinforcement resists the shear force (related to bending along the strong axis) and torsion; in most cases it corresponds to the cross-section of the vertical transverse reinforcement bars per unit beam length;

: to plot transverse reinforcement quantities parallel to the strong axis of the cross-section. This reinforcement resists the shear force (related to bending along weak axis) and torsion; in most cases it corresponds to the cross-section of the horizontal transverse reinforcement bars per unit beam length;

Cross-section check (steel & timber)

The last icons within the toolbox relate to the cross-section check for steel & timber. This verification typically involves 2 parts : while the first one relates to the strength of the cross section, the other one corresponds to the verification of the buckling stability. The icons and allow to access both types, which are always expressed as a percentage of strength/stability capacity of each member.

IMPORTANT : It is possible, in case one of both buttons is pressed down, to double-click on any member of the model in the 'Plot'-window to show a dialogue box in which more detail is provided on strength/stability verification of the selected bar. Within this dialogue box, it is also possible, if required, to modify the buckling length of the selected member. This item will be further discussed in another section of this reference manual.

3.1.11 The 'Data' window

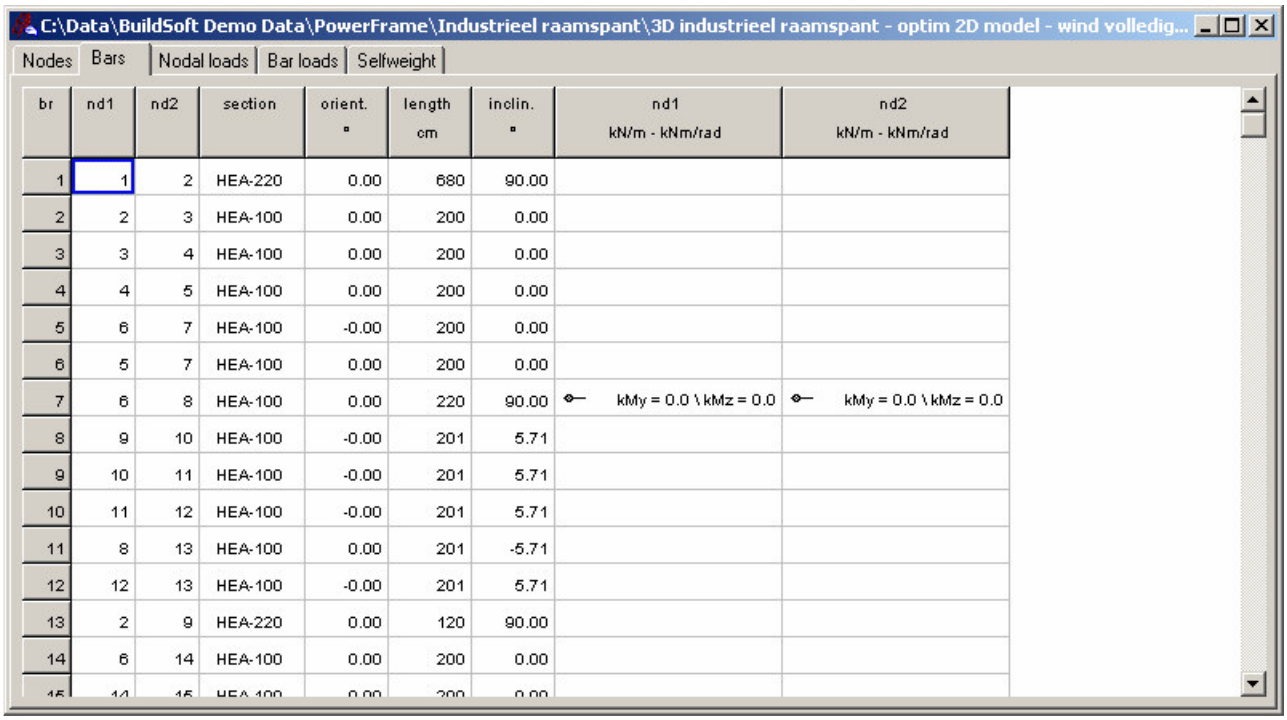
The 'Data' window consists of a number of tables containing all data describing the model. This includes for instance nodal coordinates, definition of loads & end conditions, cross-section properties, etc.

This window contains 5 tab sheets. The first one relates to the coordinates of the nodes describing the model geometry, along with specific constraints that

have been defined at nodes. The second tab sheet describes all members of the model: end node numbers, member orientation, connection at end points, etc.

The next sheet gives an overview of the loads that have been defined at nodes for the load case currently active in the 'Loads' window. The fourth tab sheet defines the loads assigned to members. Finally, the last sheet summarizes the section names, materials, lengths, selfweights per unit length, volumes, painting surface for all members. At the bottom, this table also gives the total weight and painting area of all members.

All tables display only information about the visible parts of the structural model.



br	nd1	nd2	section	orient. °	length cm	inclin. °	nd1 kN/m - kNm/rad	nd2 kN/m - kNm/rad
1	1	2	HEA-220	0.00	680	90.00		
2	2	3	HEA-100	0.00	200	0.00		
3	3	4	HEA-100	0.00	200	0.00		
4	4	5	HEA-100	0.00	200	0.00		
5	6	7	HEA-100	-0.00	200	0.00		
6	5	7	HEA-100	0.00	200	0.00		
7	6	8	HEA-100	0.00	220	90.00	kMy = 0.0 \ kMz = 0.0	kMy = 0.0 \ kMz = 0.0
8	9	10	HEA-100	-0.00	201	5.71		
9	10	11	HEA-100	-0.00	201	5.71		
10	11	12	HEA-100	-0.00	201	5.71		
11	8	13	HEA-100	0.00	201	-5.71		
12	12	13	HEA-100	-0.00	201	5.71		
13	2	9	HEA-220	0.00	120	90.00		
14	6	14	HEA-100	0.00	200	0.00		

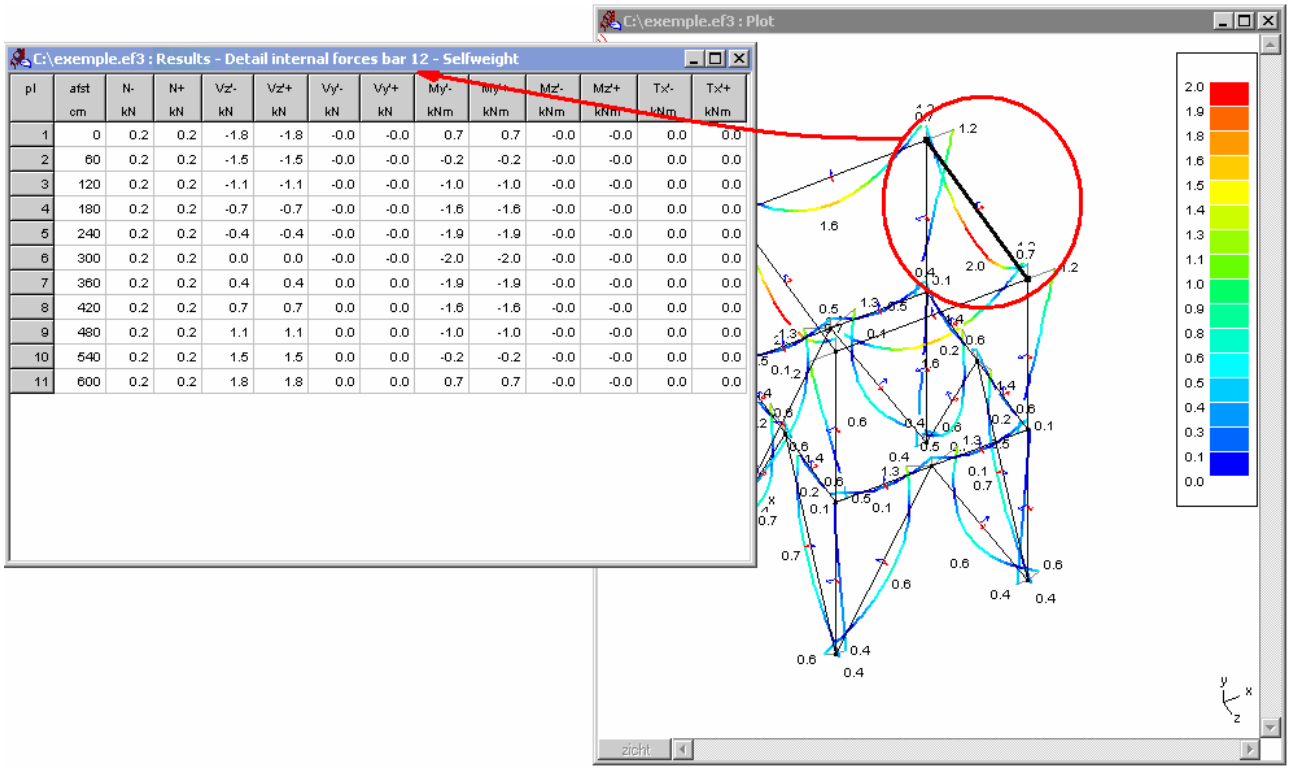
The user can modify the values contained in the sheets 'nodes', 'loads at nodes' and 'loads on members'.

Remark: All tables presented by PowerFrame can be exported to a spreadsheet tool like MS Excel or another program using the Copy/Paste capabilities of MS Windows.

3.1.12 The 'Results' window

The 'Results' window provides access to the results of the PowerFrame analysis in a tabular form. This window always operates in parallel to the 'Plot' window, which means that the results displayed in the 'Results' window are always those **actually** shown in the 'Plot' window. It should be noted

explicitly that the ‘Results’ window will only present data related to members which are visible within the ‘Plot’ window. The results that are presented in tabular form relate mostly to the end nodes of the selected members, except in the case of reinforcement quantities and cross-section strength/ buckling stability verification. In these cases, the table always gives the maximum value over the entire bar element. If only one bar is selected in the ‘Plot’ window, the ‘Results’ window will present result values at both end nodes and at 9 intermediate nodes along the member.



pl	afst cm	N- kN	N+ kN	Vz'- kN	Vz'+ kN	Vy'- kN	Vy'+ kN	My'- kNm	My'+ kNm	Mz'- kNm	Mz'+ kNm	Tx'- kNm	Tx'+ kNm
1	0	0.2	0.2	-1.8	-1.8	-0.0	-0.0	0.7	0.7	-0.0	-0.0	0.0	0.0
2	60	0.2	0.2	-1.5	-1.5	-0.0	-0.0	-0.2	-0.2	-0.0	-0.0	0.0	0.0
3	120	0.2	0.2	-1.1	-1.1	-0.0	-0.0	-1.0	-1.0	-0.0	-0.0	0.0	0.0
4	180	0.2	0.2	-0.7	-0.7	-0.0	-0.0	-1.6	-1.6	-0.0	-0.0	0.0	0.0
5	240	0.2	0.2	-0.4	-0.4	-0.0	-0.0	-1.9	-1.9	-0.0	-0.0	0.0	0.0
6	300	0.2	0.2	0.0	0.0	-0.0	-0.0	-2.0	-2.0	-0.0	-0.0	0.0	0.0
7	360	0.2	0.2	0.4	0.4	0.0	0.0	-1.9	-1.9	-0.0	-0.0	0.0	0.0
8	420	0.2	0.2	0.7	0.7	0.0	0.0	-1.6	-1.6	-0.0	-0.0	0.0	0.0
9	480	0.2	0.2	1.1	1.1	0.0	0.0	-1.0	-1.0	-0.0	-0.0	0.0	0.0
10	540	0.2	0.2	1.5	1.5	0.0	0.0	-0.2	-0.2	-0.0	-0.0	0.0	0.0
11	600	0.2	0.2	1.8	1.8	0.0	0.0	0.7	0.7	-0.0	-0.0	0.0	0.0

Note : Just like all other tables presented by PowerFrame, the table of results can be exported to a spreadsheet tool like MS Excel or another program by using the Copy/Paste possibilities of MS Windows.

3.2 Calculation of buckling lengths

PowerFrame calculates the buckling length of individual members. Consider the example of a member which is part of a structural model, for which the user wants to calculate the buckling length with respect to the strong and weak axis. The buckling length calculation starts with the application of a uniformly distributed unit load on the member, in the direction for which the buckling length is to be calculated. The response of the whole structural model to this load is then calculated. The following specific results are of interest:

- displacement (v) and rotation (φ) at both end points, related to the direction in which the load is applied.
- bending moment (M) and shear force (V) at both end points, again related to the direction in which the load is applied

PowerFrame will then determine the ratios V/v and M/φ to obtain values for the translational and rotational stiffness which the bar encounters at its end points where it connects to the remaining part of the structure. Using those stiffness values (which can be considered as the equivalent stiffness of springs connected to the isolated bar), the buckling length of the isolated bar can be calculated using the Euler differential equation

$$EIv''''+Pv''=0$$

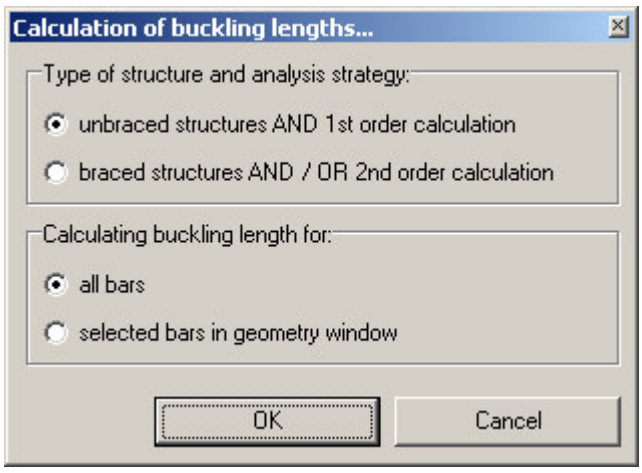
of which the general solution is

$$v = A\sin\alpha x + B\cos\alpha x + Cx + D \quad avec\ \alpha = \sqrt{\frac{P}{EI}}$$

By imposing the appropriate boundary conditions at both end points (using the previously derived spring stiffness values), a system of 4 equations with 4 unknowns is obtained. Looking for non-trivial solutions of this system of equations, several possible values for α can possibly be obtained. The smallest value of α corresponds to the buckling load of the member, from which the buckling length can then be derived.

PowerFrame thus performs 2 structural analyses for each bar that is part of the model, each analysis corresponding to one of 2 orthogonal directions for which buckling is to be investigated.

As a function of the type of structure (braced vs unbraced) and the type of analysis (1st order vs 2nd order), PowerFrame will use the appropriate calculation scheme. The calculation is launched through the icon .


Calculation of buckling lengths...
Type of structure and analysis strategy:
unbraced structures AND 1st order calculation
braced structures AND / OR 2nd order calculation
Calculating buckling length for:
all bars
selected bars in geometry window
OK
Cancel

If buckling lengths are to be evaluated for members that belong to a structural model which is calculated at the 2nd order, or which can be considered as a braced structure, design standards allow to evaluate the buckling length of a bar assuming the end nodes of the bar do not move horizontally. In this case, PowerFrame will assume the translation DOFs of the model to be fixed. On the other hand, if the structure is to be considered as unbraced and if a 1st order analysis is performed, PowerFrame will not use this assumption of fixed nodes, but will evaluate the translational stiffness at nodes as previously described.

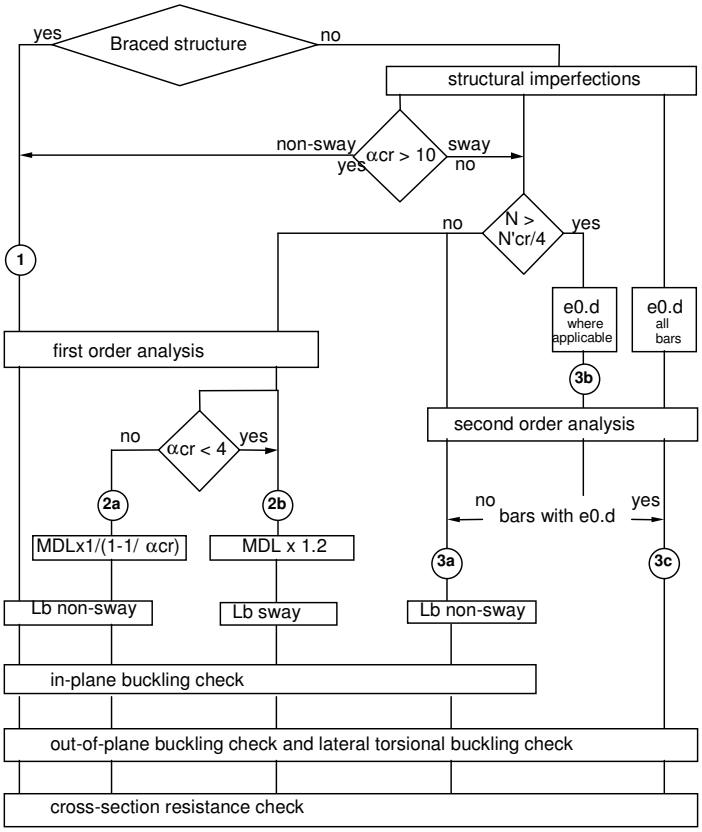
To visualize the buckling length of the members directly on the model, use the main menu 'Window – General parameters' and make the required selections in the dialogue window.

To modify the buckling length of a particular bar, first click on the lower RHS button of the icon toolbox (buckling check) in case of a structure containing steel and/or timber members, or on one of the reinforcement buttons in case of a structure including concrete members. Once the appropriate icon has been used, simply double-click on a member to show and possibly modify the buckling length for 2 orthogonal directions.

3.3 Design analysis

3.3.1 Static analysis

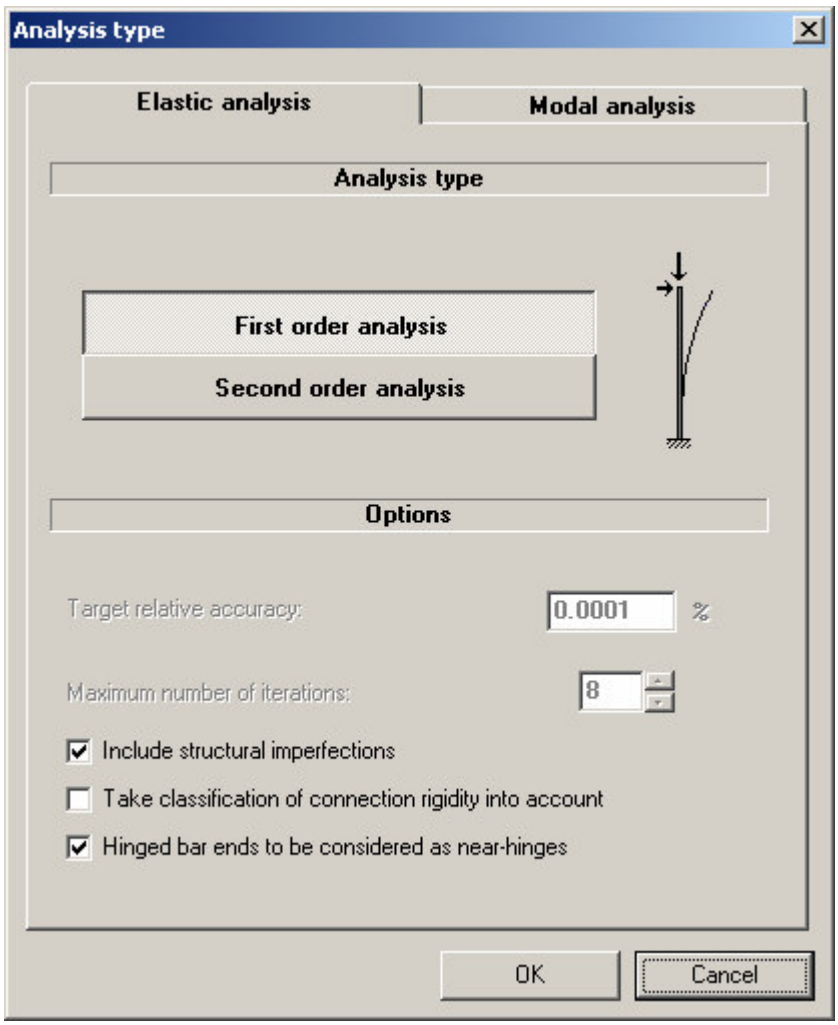
PowerFrame supports multiple global analysis strategies. Prior to explaining these in detail, gain of optimal and exhaustive understanding of the scope and context of each strategy is advisable.



αcr : critical load factor (vertical loads global elastic buckling of structure)
N : axial force in bar (design value ULS)
N'cr : critical axial load of bar based on system length
$e_{0.d}$: bar imperfection
MDL : part of bending moments related to horizontal displacements, see 5.2.6.2(5)
Lb : buckling length – to be used in buckling check

Refer to the flow chart above on possible global analysis schemes as specified by Eurocode 3. Navigating through the flow chart, one of the first questions to be answered relates to the sensitivity of the structure to horizontal loads. If a structure can be considered to be braced, in other words, if a correctly designed bracing system is capable of transmitting horizontal forces to the supports of the structure, it can be assumed that the structure presents sufficient stiffness to neglect the impact of 'global imperfections'. In this case, a second order analysis is not required. This corresponds to path 1 in the above flow-chart. Nevertheless, this will be a relatively rare situation. A more refined analysis strategy (corresponding to one of the other paths in the flow chart) will mostly be required.

PowerFrame does not automatically navigate through the above flow-chart. The user has to evaluate which conditions are met and which ones are not met and choose the appropriate analysis strategy to use in the dialogue shown below. This dialogue is activated using the icon.



When a structure cannot be considered to be braced, global structural imperfections need to be accounted for by selecting the appropriate option in the dialogue window. Depending on the outcome of all checks in the above flow chart, the user has to select the proper analysis strategy in the dialogue window. Note that this choice also affects the strategy for the calculation of buckling lengths for individual members, as follows:

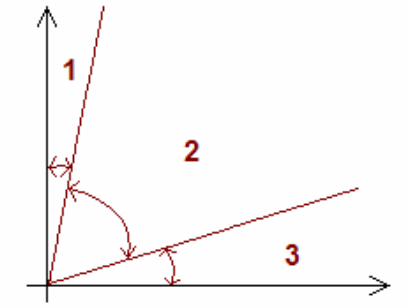
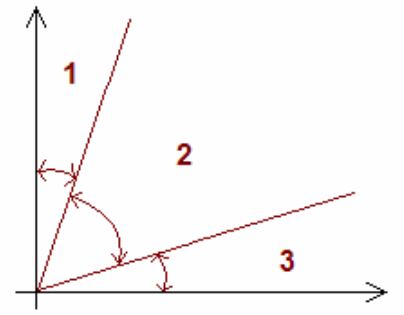
Path in flow-chart	Buckling length	1st or 2nd order analysis	Take into account global imperfections
1	Non-sway mode	1st order	No
2b	Sway mode	1st order	Yes
3a	Non-sway mode	2nd order	Yes

The last option in the dialogue window allows to specify whether the hinged connections in the model need to be considered as « perfectly hinged » (having a truly zero rotational stiffness) or if a small finite rotational stiffness should be assigned to them (“near-hinges”). In case part of a structure is not properly connected to the rest of the structure and actually behaves as a

mechanism, the introduction of near-hinges does allow PowerFrame to calculate the structure and hence also to present the analysis results and visualize possible mechanisms that exist in the model. As a result, this option can offer a high added value to the user as a diagnostic tool.

One more item remains to be examined: the classification of connection rigidity. This feature is particularly useful with PowerFrame Master. The actual version of PowerFrame allows for a bi-directional interaction between the global frame analysis and the detail design of the connection between individual members of the structural frame.

The detail analysis of the connection will not only evaluate the actual stiffness of the connection, but will also classify the node as rigid, semi-rigid or hinged, depending on the actual stiffness value as compared to the limit values (see below).

Unbraced structure	**Braced structure**
	
Zone 1 : rigid (Sj,ini ≥ 25 E Ib / Lb)	Zone 1 : rigid (Sj,ini ≥ 8 E Ib / Lb)
Zone 2 : semi-rigid	Zone 2 : semi-rigid
Zone 3 : hinged (Sj,ini ≤ 0,5 E Ib / Lb)	Zone 3 : hinged (Sj,ini ≤ 0,5 E Ib / Lb)

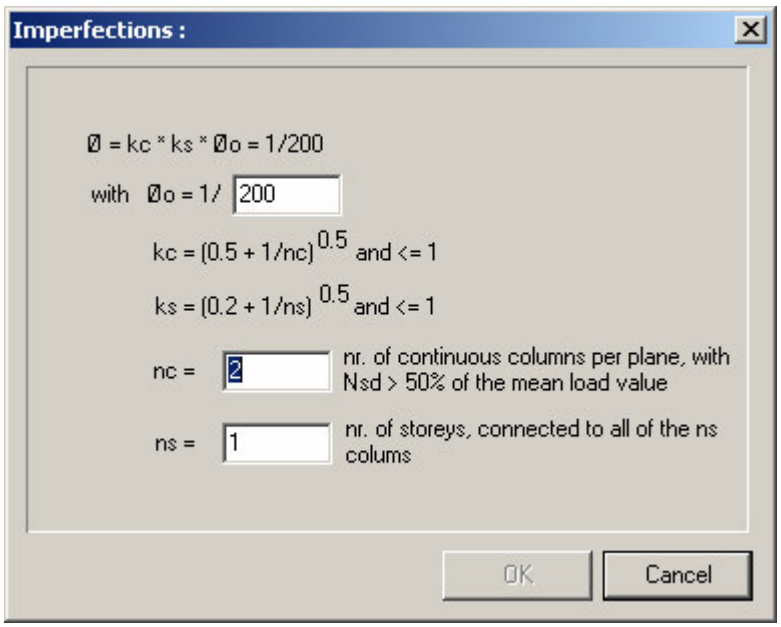
If the user wants to consider the rigid connections as truly rigid and hinged connections as perfectly hinged, he should select, to this end, the corresponding option in the dialogue window. If, on the other hand, he wants to use the real stiffness value of the connection during the analysis, this option should remain unselected.

Note: The rigidity classification of a connection is saved in the PowerFrame project as part of the stiffness properties defined at member ends. To access this information, go to the 'Geometry' window and click on the icon which allows to define hinges at bar ends.

3.3.2 Global structural imperfections

As already mentioned in the previous section, PowerFrame can take into account global structural imperfections. This is done through the introduction of initial sway imperfections which will generate secondary forces into the structure similar to those that are inevitably introduced during the construction of the building.

All parameters which are necessary to properly define those initial sway imperfections are defined through the menu entry 'Analysis' - 'Imperfections…'.



The dialogue window that appears contains the formula used to define the horizontal deviation of the structure with respect to its undeformed initial position in such a way that it allows for a modelling of global imperfections. It is based directly on Eurocode 3 and uses a reference value of 1/200 multiplied by coefficients kc and ks that depend on the number of most-loaded columns per building level (nc) and the number of building levels (ns). Both parameters should be specified by the user.

3.3.3 Steel & timber design analysis

Once the global elastic analysis has been performed, deformations and internal forces are available as analysis results. For members made of another material than concrete, elastic stresses are also calculated during the analysis. For steel or timber members, 2 major additional checks can be made:

- verification of the cross-section strength
- verification of the buckling stability.

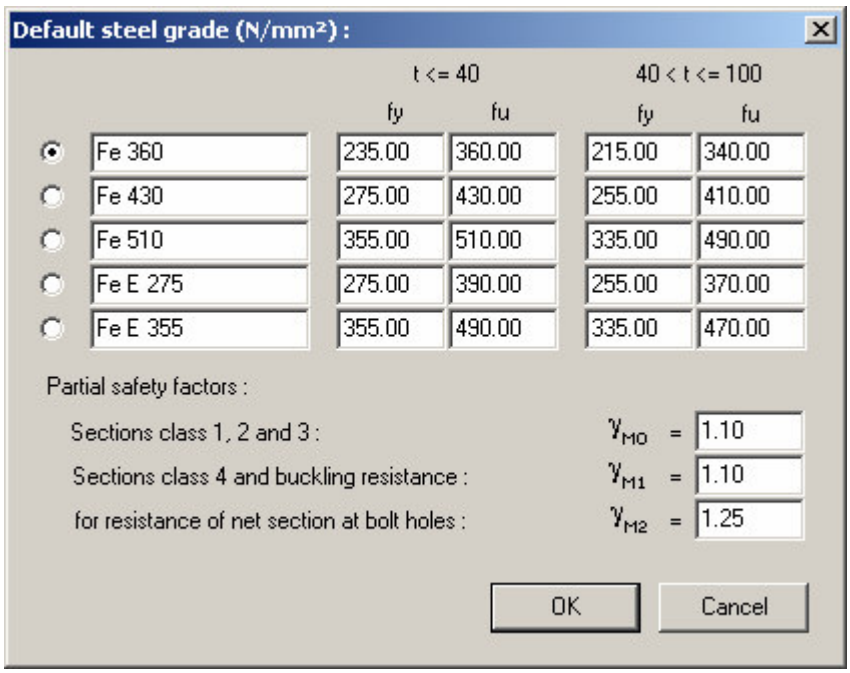
3.3.3.1 Selection of the design code

Verification of timber members is always performed in accordance with Eurocode 5, whereas a wide range of design codes is available for the verification of steel members, like :

The appropriate design code can easily be selected through the menu entry ‘Analysis’ – ‘Steel design code’.

3.3.3.2 Steel design parameters

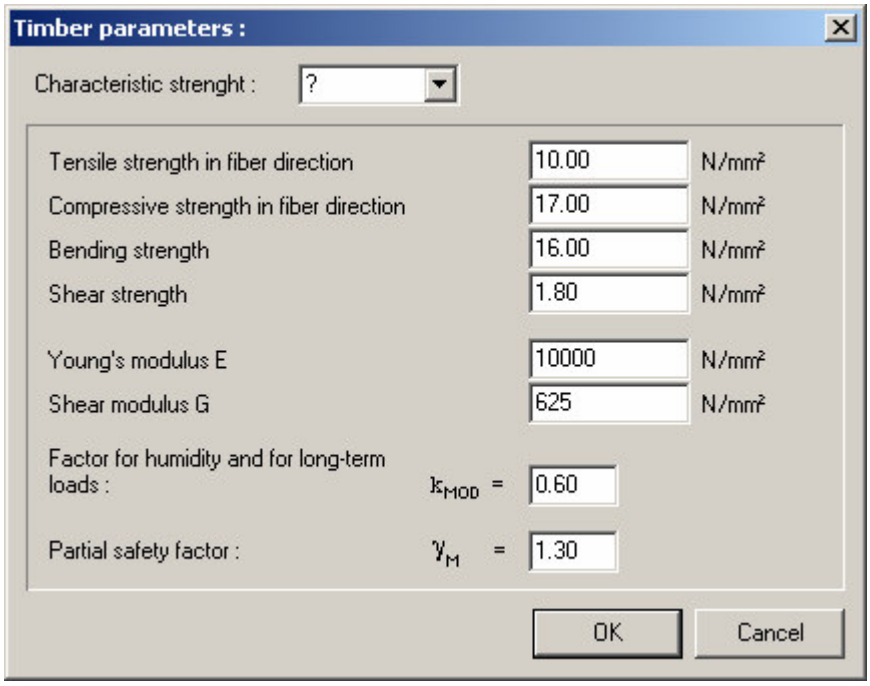
To define the material properties that affect the design code check, select the menu entry ‘Analysis’ – ‘Steel design parameters…’. In the dialogue window that appears, 5 different steel grades can be selected. One of these can be selected as default steel grade, which will automatically be assigned to all newly defined steel members. Next to the relevant strength characteristics, PowerFrame also asks the user to specify the partial safety factors γ_{M0}, γ_{M1} and γ_{M2}. By default, those factors equal 1.1, 1.1 and 1.25, which are the values proposed by Eurocode 3. Deviations from those values can be defined for each country and are listed in a national application document (NAD). Some standards do not use those partial safety coefficients at all (as for instance, the French standard CM66): in this case, all verifications are done with all partial safety coefficients equal to 1.



Please note that names of the different steel grades can freely be chosen by the user, as well as the corresponding material properties. The default steel grade is defined by selecting the appropriate radio button.

3.3.3.3 Timber design parameters

To define the parameters to be used for the timber design analysis, the user should select the menu entry ‘Analysis – Timber design parameters...’. and will then have access to the dialogue window shown below :



Through a pull-down menu in the upper part of the dialogue window, the user can select the appropriate timber class. The corresponding material properties are filled out automatically in the editor fields. The values of De k_{MOD} and γ_M remain unaffected when a different timber class is selected.

3.3.3.4 Verification of the cross-section strength

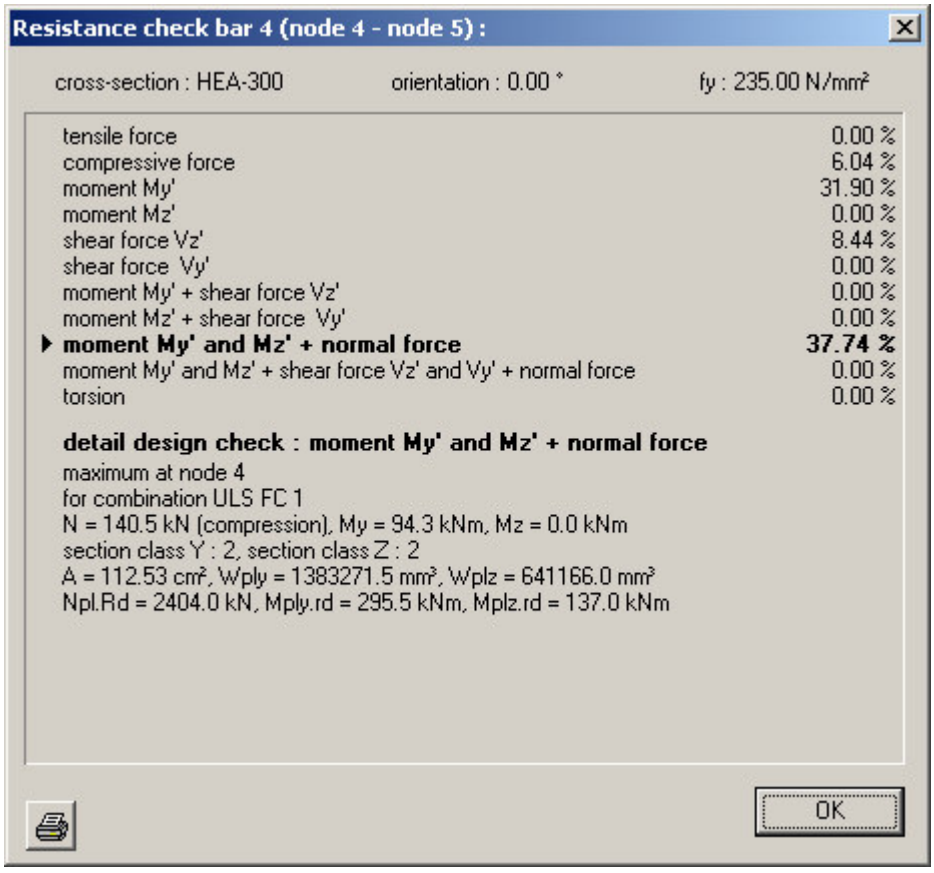
During the verification of the cross-section strength, PowerFrame checks whether the calculated internal forces exceed the design values specified by the selected standard. This verification takes into account the appropriate safety coefficients on material properties and loads, as previously defined by the user. For most of the design codes, this verification is to be performed for the ultimate limit states. PowerFrame performs this verification for all load combinations in ultimate limit states.

During this verification process, several checks are performed. In case Eurocode 3 has been selected as design standard, the following checks are made :

- Tensile force ;
- Compressive force ;
- Bending moment My' ;
- Bending moment Mz' ;
- Shear force Vy' ;
- Shear force Vz' ;
- Bending moment My' combined with shear force Vz' ;
- Bending moment Mz' combined with shear force Vy' ;
- Bending moments My' and Mz' combined with axial force ;
- Bending moments My' and Mz' combined with axial - and shear forces Vy' and Vz' ;
- Torsional moment T.

Verification results are always expressed as a percentage of the member's design resistance for the (combination of) internal forces under consideration.

To view those results, use the icon of the toolbox in the Plot-window. In case more details are required concerning the verification results for a specific member, double click on the member in the Plot-window.


Resistance check bar 4 (node 4 - node 5) :
cross-section : HEA-300
orientation : 0.00 °
fy : 235.00 N/mm²
tensile force 0.00 %
compressive force 6.04 %
moment My' 31.90 %
moment Mz' 0.00 %
shear force Vz' 8.44 %
shear force Vy' 0.00 %
moment My' + shear force Vz' 0.00 %
moment Mz' + shear force Vy' 0.00 %
moment My' and Mz' + normal force 37.74 %
moment My' and Mz' + shear force Vz' and Vy' + normal force 0.00 %
torsion 0.00 %
detail design check : moment My' and Mz' + normal force
maximum at node 4
for combination ULS FC 1
N = 140.5 kN (compression), My = 94.3 kNm, Mz = 0.0 kNm
section class Y : 2, section class Z : 2
A = 112.53 cm², Wply = 1383271.5 mm³, Wplz = 641166.0 mm³
Npl.Rd = 2404.0 kN, Mply.rd = 295.5 kNm, Mplz.rd = 137.0 kNm
OK

In the dialogue window, all verifications are summarized (in the above example, all verifications have been performed based on Eurocode 3). The specific design check which yields the highest percentage of design resistance, is always indicated in bold. For this specific design check, the detail information is presented in bold in the lower half of the window. To access detail information on the other design checks, the user selects the appropriate design check with the mouse and the requested information will automatically appear in the lower half of the window.

3.3.3.5 Verification of the buckling stability

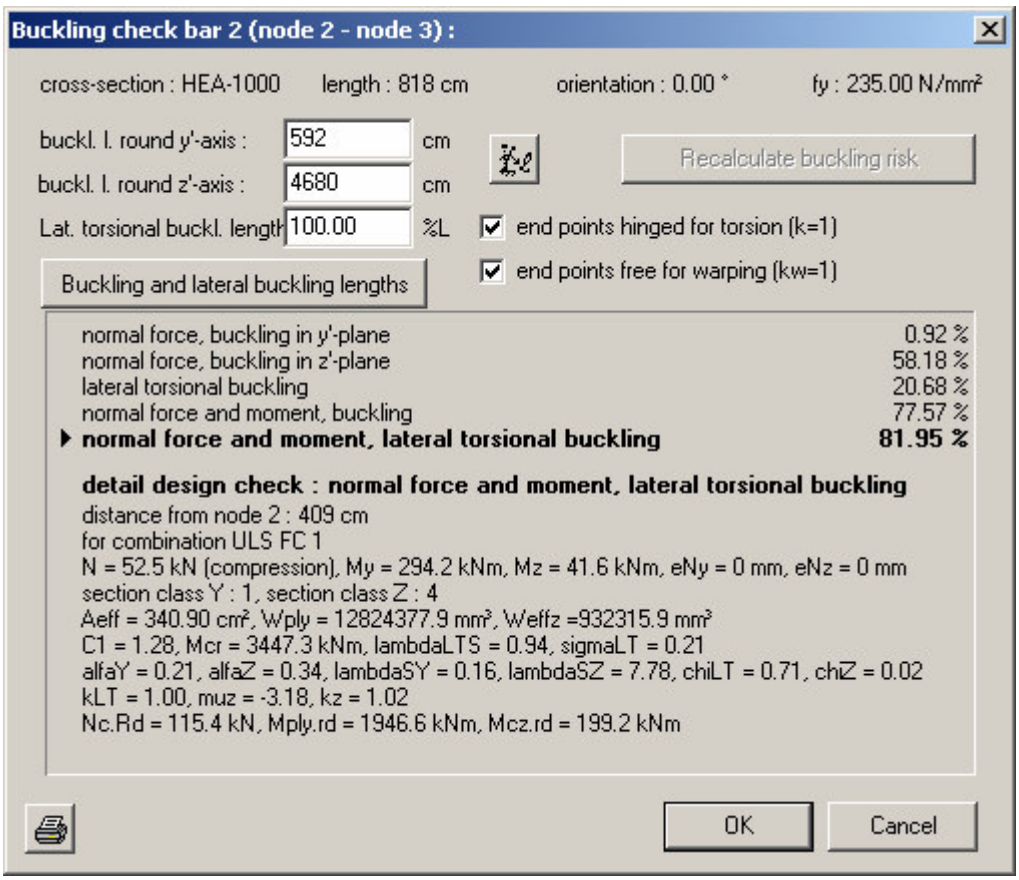
To access the results of the buckling stability check as a percentage of the buckling stability design value, select the icon from the toolbox with the 'Plot- window..

Completely similar to the presentation of cross-section resistance verification results, several checks are performed as part of buckling stability verification. In case of Eurocode 3, following checks are made:

- compression force – buckling with respect to strong axis of cross-section;

- compression force – buckling with respect to weak axis of cross-section;
- lateral torsional buckling ;
- buckling in presence of compression force and bending moment ;
- lateral torsional buckling, in presence of compression force and bending moment.

To get more details on the buckling stability check of a particular structural member, just double-click on this member in the 'Plot'-window.


Buckling check bar 2 (node 2 - node 3) :
cross-section : HEA-1000
length : 818 cm
orientation : 0.00 °
fy : 235.00 N/mm²
buckl. l. round y'-axis :
592
cm
buckl. l. round z'-axis :
4680
cm
Lat. torsional buckl. length
100.00
%L
Recalculate buckling risk
end points hinged for torsion (k=1)
end points free for warping (kw=1)
Buckling and lateral buckling lengths
normal force, buckling in y'-plane 0.92 %
normal force, buckling in z'-plane 58.18 %
lateral torsional buckling 20.68 %
normal force and moment, buckling 77.57 %
normal force and moment, lateral torsional buckling 81.95 %
detail design check : normal force and moment, lateral torsional buckling
distance from node 2 : 409 cm
for combination ULS FC 1
N = 52.5 kN (compression), My = 294.2 kNm, Mz = 41.6 kNm, eNy = 0 mm, eNz = 0 mm
section class Y : 1, section class Z : 4
Aeff = 340.90 cm², Wply = 12824377.9 mm³, Weffz =932315.9 mm³
C1 = 1.28, Mcr = 3447.3 kNm, lambdaLTS = 0.94, sigmaLT = 0.21
alfaY = 0.21, alfaZ = 0.34, lambdaSY = 0.16, lambdaSZ = 7.78, chiLT = 0.71, chiZ = 0.02
kLT = 1.00, muz = -3.18, kz = 1.02
Nc.Rd = 115.4 kN, Mply.rd = 1946.6 kNm, Mcz.rd = 199.2 kNm
OK
Cancel

Independent of the selected design standard, the buckling length of the member can be changed by directly editing the appropriate fields in the above dialogue window, or by using the icon . Once the buckling length has been changed, a new buckling stability check can be performed for the member by using the button 'Recalculate buckling risk'.

When performing a buckling stability verification with respect to Eurocode 3, the possibility exists to consider the presence of lateral torsional stiffeners. Those stiffeners do reduce the lateral torsional buckling length to values which can be significantly smaller than the member's system length. For instance, it can be seen that when using 3 lateral torsional stiffeners, the lateral torsional buckling length of the member is reduced to ¼ of the system

length when the stiffeners are positioned at a relative distance of ¼ of the system length.

Two additional options are available when performing the torsional buckling check :

- end points hinged for torsion (k = 1)
- end points free for warping (kw = 1)

If the user has selected the option ‘End points hinged for torsion’, the member will be able to rotate around its axis when subjected to a torsional moment at the end points. It is clear that such boundary conditions significantly reduce the resistance against lateral torsional buckling. If boundary conditions at the end points of the member are such that the above described rotation is restrained, this option can be unselected.

In case the second option ‘End points free for warping‘ is selected, the end sections will not be constrained to remain plane. Again, this type of boundary condition reduces the resistance against lateral torsional buckling. For cross-sections which are highly sensitive to warping, warping of the end sections can be strongly reduced through the use of welded end plates. In this case, the option ‘End points free for warping’ can be unselected.

3.3.3.6 Cross-section optimization

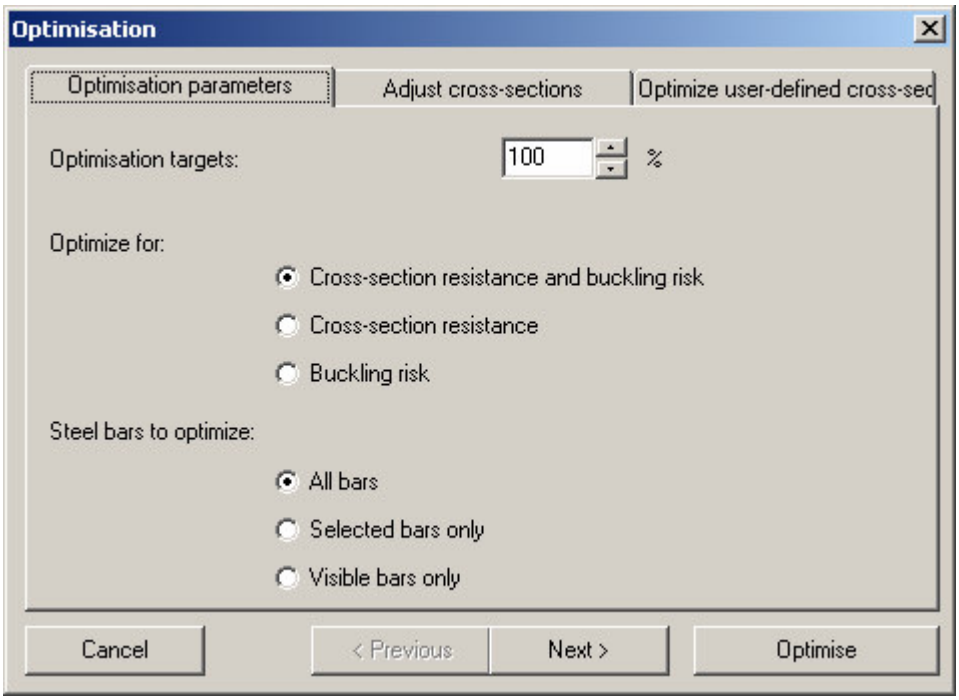
The verification of cross-section resistance and member buckling stability delivers a result expressed as a percentage of the member’s design resistance for the (combination of) internal forces under consideration. An optimal dimension is realized when those results are as near to 100% as possible, without exceeding this target value however.

PowerFrame includes optimization capabilities which allow to determine the most optimal cross-section for the internal forces under consideration.

Optimization is achieved by a variation of cross-section properties according to one of the following principles:

- for cross-sections selected from the cross-section library, the optimization procedure looks for a cross-section within the same group (HEA, IPE, …) as the original cross-section which approaches the desired target value as close as possible.
- for cross-sections defined on cross-section types, the optimization procedure will search for optimal cross-section dimensions by modifying height, width, web or flange thickness (depending on the preferences imposed by the user).

To start the optimization procedure, click on the Icon. The next dialogue window appears:



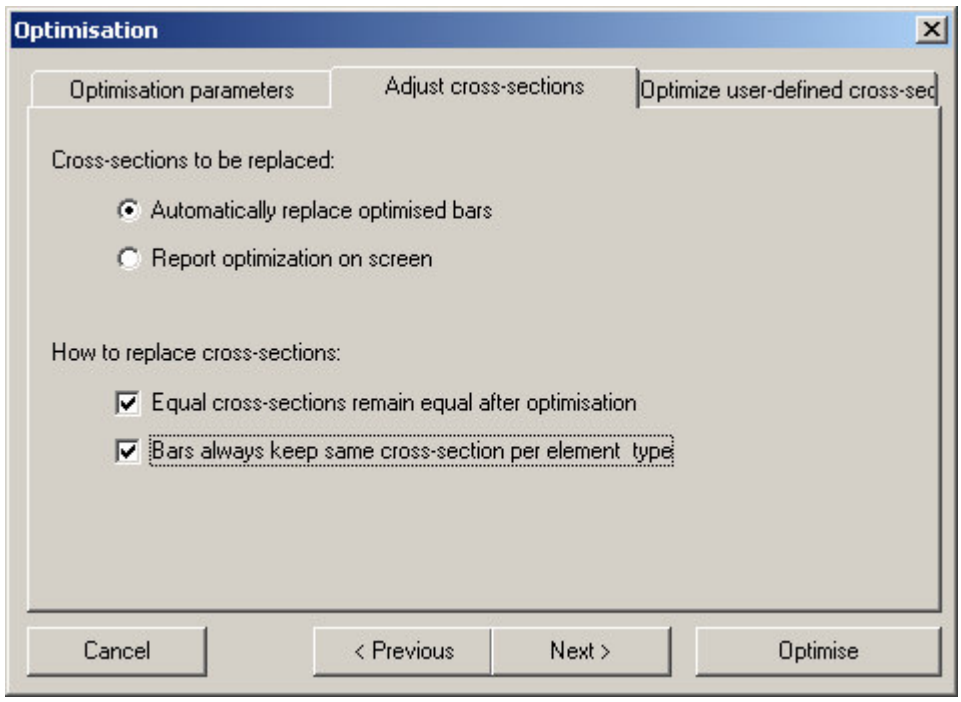
The first tab page allows to specify the optimization targets. Indicate whether the optimization must be realized:

- for both cross-section resistance and member buckling risk
- for cross-section resistance only
- for member buckling risk only

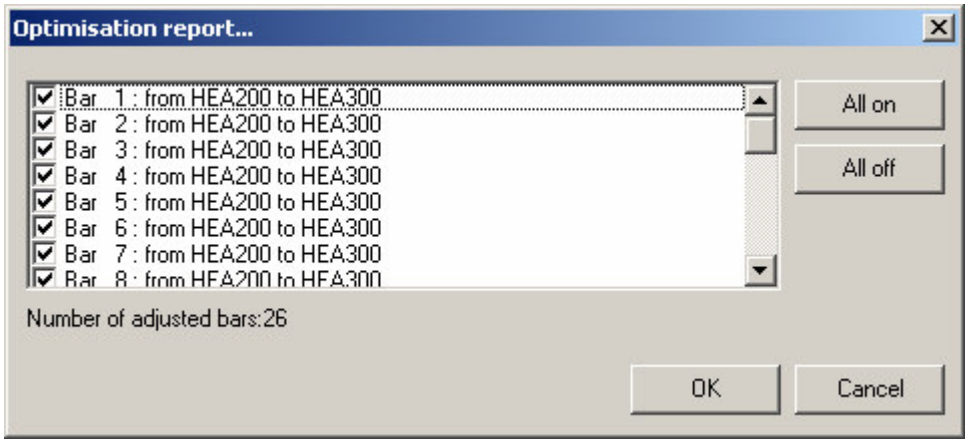
The optimization can be done for:

- all bars
- those bars which are selected in the ‘Plot’ window
- those bars which are visible

The second tab page concerns all parameters relative to the optimization of cross-sections selected from the cross-section library.



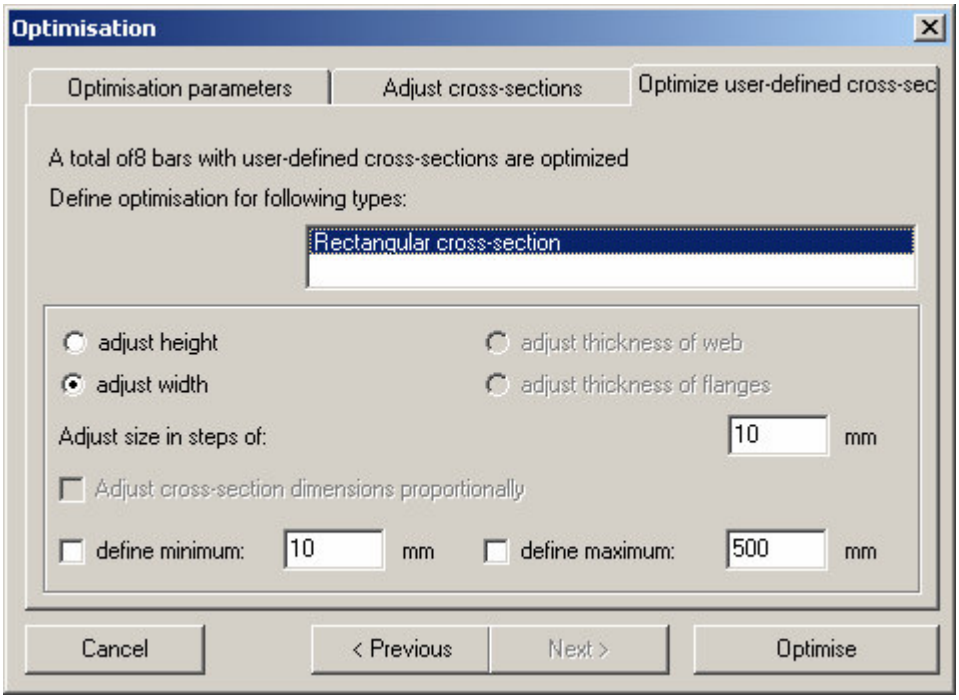
By selecting the first option, cross-sections will automatically be replaced for optimized bars. In the other case, a summary will appear at the end of the optimization procedure and the user will be asked to confirm or reject the proposed cross-section.


Optimisation report...
Bar 1 : from HEA200 to HEA300
Bar 2 : from HEA200 to HEA300
Bar 3 : from HEA200 to HEA300
Bar 4 : from HEA200 to HEA300
Bar 5 : from HEA200 to HEA300
Bar 6 : from HEA200 to HEA300
Bar 7 : from HEA200 to HEA300
Bar 8 : from HEA200 to HEA300
All on
All off
Number of adjusted bars:26
OK
Cancel

Two additional options are available with the optimization functionality:

- the option ‘Equal cross-sections remain equal after optimization’ assures that all bars which did have identical cross-sections before the optimization started, will still have identical cross-sections after optimization.
- the second option will change all cross-sections corresponding to one element type into one single optimal cross-section.

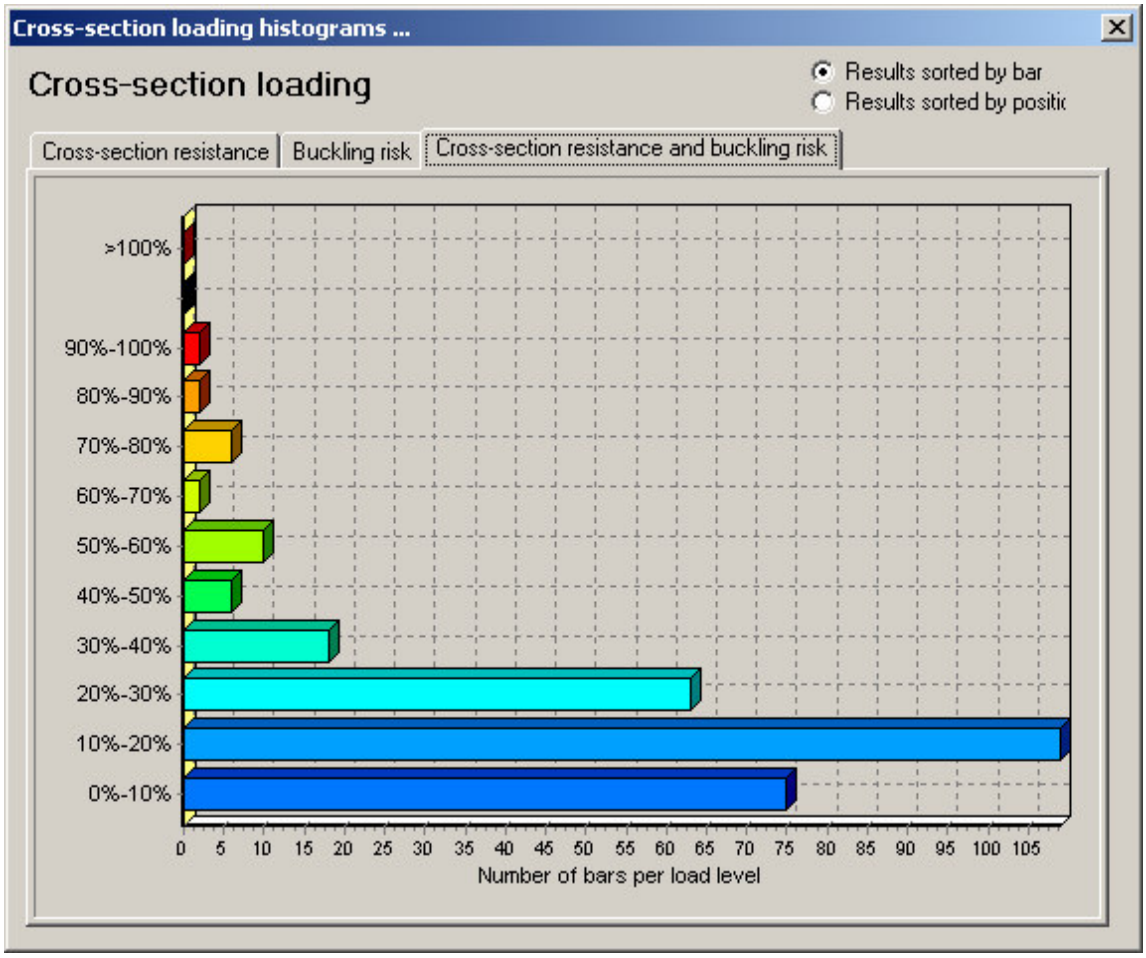
Finally, the third tab page concerns all parameters related to the optimization of cross-sections defined on cross-section types.



First of all, PowerFrame presents the total number of bars with user-defined cross-sections that can be optimized. Next, all available cross-section types are shown in a list. After selecting a cross-section type, extra information will appear in the lower half of the dialogue window, allowing to define the optimization parameters.

3.3.3.7 Loads histogram

Once the cross-section resistance and buckling stability verification have been performed, the « load capacity » of all bars can be summarized graphically. For each member, the load capacity is evaluated based on the verification results in 11 intermediate points. To visualize this graphical summary, use the icon .



Along the horizontal axis, the number of member or the number of evaluation points will be indicated, depending on which option has been selected ("Results sorted by bar" or "Results sorted by position"). Along the vertical axis, the "load capacity" is presented in intervals of 10% each.

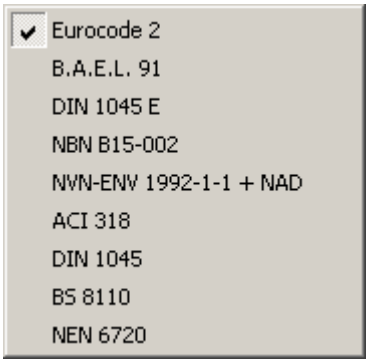
Histograms can be visualized based either on the cross-section verification only, or on the buckling stability verification only, or on a combination of both. The higher the number of members in the upper region (approaching the target value of 100%), the more efficient use will be made of all structural members. Of course, the limit value of 100% should not be exceeded for any member – at least, in principle.

3.3.4 Calculation of reinforcement quantities

3.3.4.1 Selection of R.C. design code

For those members which have been assigned a material property of type 'concrete', PowerFrame can further use the results of an elastic analysis to calculate the required reinforcement quantities.

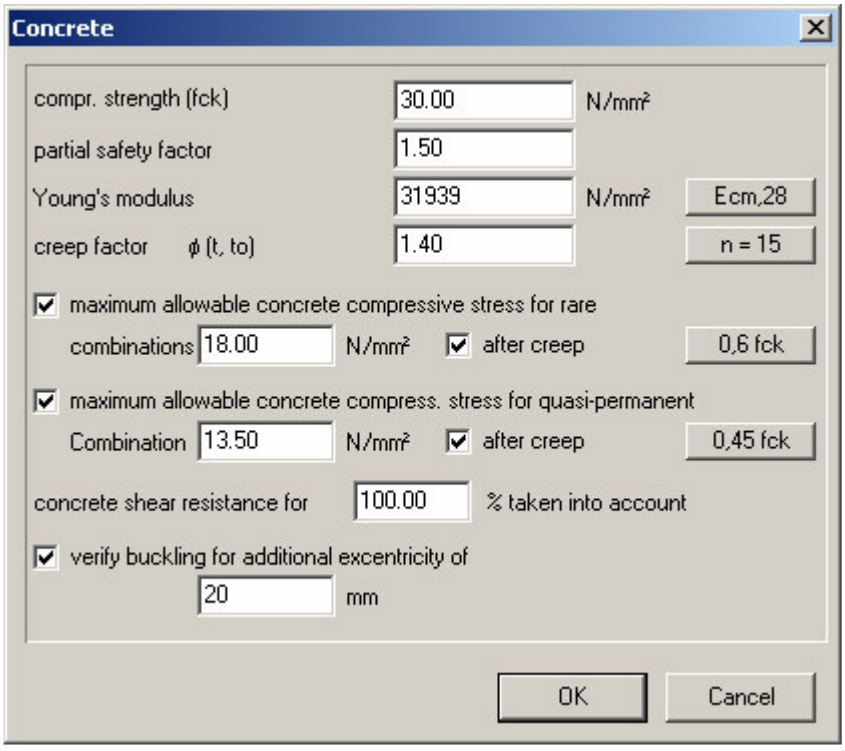
Results of this calculation can be slightly different depending on the selected design code. This selection can be made through the main menu, by going to the menu entry ‘Analysis – R.C. design code’. Following design codes are currently supported :



Depending on the selected design code, a number of material properties - needed for the evaluation of reinforcement quantities and further organic calculations – need to be defined.

3.3.4.2 Concrete parameters

The concrete properties can be specified through the menu entry ‘Analysis – R.C. design parameters – Concrete…’. The following dialogue window relates to Eurocode 2:



The characteristic compressive strength f_{ck} is evaluated on test cylinders of size 150 by 300 mm, at an age of 28 days. The partial safety factor mostly equals 1.50

Note : despite the fact that the Young's modulus of concrete has been defined as part of the materials library, this dialogue window also requires a specification for this characteristics. It is important to remember that both moduli are used for different types of analysis:

- *Young's modulus as defined in the materials library is used for the elastic analysis of the structural model. It helps to evaluate the elastic stiffness of the structure and to calculate elastic deformations and internal forces*
- *Young's modulus as defined in the above dialogue window is used exclusively for the so-called organic calculations. It is used in the concrete stresses computation, based on the results of the previous elastic analysis. The button '$E_{cm,28}$' in the dialogue above evaluates the secans modulus at the age of 28 days, resulting from the characteristic compressive strength f_{ck} .*

The creep factor φ(t,t0) can be specified directly by the user or be calculated so that the ratio of Young's modulus of reinforcement steel, E_S = 200.000 N/mm², to the Young's modulus of concrete, including all creep effects ($E_C = \frac{E_{C,28}}{1+\varphi}$), equals 15:

$$15 = \frac{E_S}{\frac{E_{C,28}}{1+\varphi}}$$

This comes down to calculating the creep factor with the formula below:

$$\varphi = \frac{15 \times E_{C,28} - E_S}{E_S}$$

in which E_s = 200 000 N/mm^2.

The next two entries in the dialogue window allow to limit concrete stresses in serviceability limit states (SLS). Again, the user can either specify the maximum allowable stresses manually or have them calculated automatically based on the recommendations of the selected design code. As concrete stresses decrease as a result of increasing creep and because the structure mostly reaches its maximum loading only after the creep effects related to permanent loads have stabilized, it is common practice to limit the concrete

stresses considering a ratio of 15 as indicated above. This method requires the following steps with PowerFrame:

- Evaluate the creep factor φ based on a Young's modulus ratio of 15
- Select the option "after creep" with the evaluation of concrete stresses

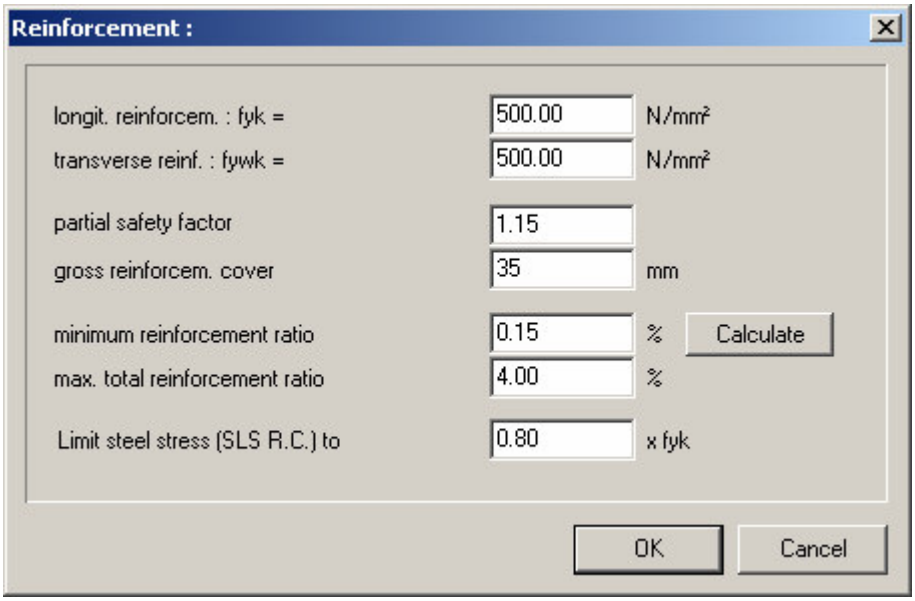
To account for a possibly reduced shear resistance at …, it is possible to limit the contribution of the concrete shear resistance to a certain percentage of the maximum concrete shear strength.

Finally, the user can specify an additional eccentricity for the verification of the buckling resistance. When this option has been selected, PowerFrame will possibly increase the calculated reinforcement quantities as to avoid buckling problems with compressed members. The verification of buckling risk is based on the "model column method", which requires the specification of an additional eccentricity to account for possible global imperfections. The user can choose between two methods :

- either perform the global elastic analysis without considering any global structural imperfections. In this case, an acceptable and reasonable value of the additional eccentricity should be specified.
- or perform the global elastic analysis taking into account the effect of global structural imperfections. In this case, a very small value for the additional eccentricity is sufficient (zero values are not accepted).

3.3.4.3 Reinforcement parameters

Next to the definition of the concrete properties required for an organic calculation, reinforcement specificiations should also be provided. This can be done through the main menu 'Analysis – R.C. design parameters – Reinforcement…' which will give access to the dialogue window below.



Note that PowerFrame allows to use different steel grades for longitudinal and transverse reinforcement.

The gross reinforcement cover corresponds to the distance between the C.O.G. of the reinforcement bars and the lower fiber of the concrete cross-section.

Minimum and maximum reinforcement ratios always relate to the geometric reinforcement ratio $\rho = A_s / b.d$. In this formula, A_s represents the total reinforcement section, while b and d correspond to the width and effective height of the concrete cross-section. The effective height d is equal to the total height h, reduced by the gross concrete cover. Whenever PowerFrame has calculated a reinforcement quantity (at bottom, top, left-hand **or** right-hand side fiber) which is lower than the minimum reinforcement ratio, the calculated quantity will have to be increased to meet this specified minimum ratio. On the other hand, the maximum reinforcement ratio always applies to the TOTAL reinforcement quantities (at bottom, top, left-hand **and** right-hand side fiber).

Finally, the steel stresses can be limited to values lower than 80% of the yield stress, as proposed by Eurocode 2. Especially for constructions in which crack width is relatively important, this reduction of steel stress can contribute to significantly lower crack widths.

3.3.4.4 Organic calculations

This section of the reference manual will NOT deal with the theoretical background of organic calculations. Instead, reference is made to Eurocode 2 and the National Standards which are supported by PowerFrame.

The organic calculation can be started as soon as the results of a global elastic analysis are available. Remember that such a global elastic analysis is always based on the material properties as defined in the materials library. The deformations calculated by this analysis are elastic deformations, without considering any effects of cracking, shrinkage or creep.

To launch the organic calculation, three possibilities are available:

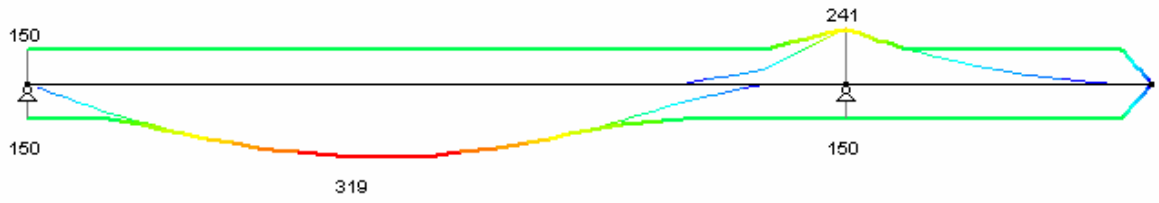
- use the icon ;
- use the menu entry 'Analysis – R.C. design analysis' in the main window ;
- use the F2 function key on the keyboard .

A dialogue window reports on calculation progress.

Once the calculation has been completed, 4 additional icons become available in the icon toolbox of the 'Plot'-window:

- shows the longitudinal reinforcement quantities parallel to the strong axis of the cross-section; in most cases, this corresponds to the upper and lower longitudinal reinforcement
- shows longitudinal reinforcement quantities parallel to the weak axis of the cross-section ; in most cases, this corresponds to the front and rear longitudinal reinforcement.
- shows transverse reinforcement quantities parallel to the weak axis of the cross-section ; this reinforcement resists torsion and shear forces corresponding to bending along the strong axis.
- shows transverse reinforcement quantities parallel to the strong axis of the cross-section ; this reinforcement resists torsion and shear forces corresponding to bending along the weak axis.

Below, the user will find a practical illustration of the upper and lower longitudinal reinforcement quantities for a simply supported beam.



In the above diagram, the thin lines correspond to the reinforcement quantities which are strictly required to comply with the ultimate limit states

(ULS) requirements. In case additional reinforcement is necessary to also comply with serviceability limit states requirements (such as limits on steel & concrete stress, minimum reinforcement ratio, consideration of buckling risk for compressed members, …), this is indicated by thicker lines. In case both line types coincide, compliance with SLS requirements does not require additional reinforcement over compliance with ULS requirements.

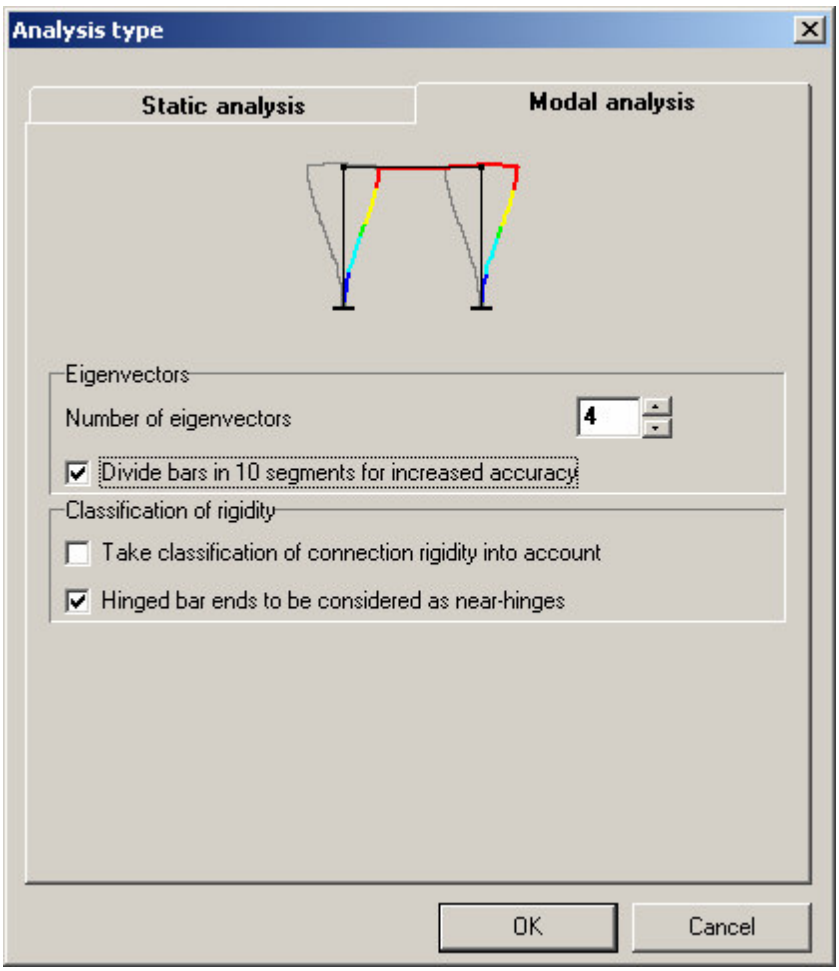
The case when the gross cross-section dimensions are insufficient to meet the theoretical reinforcement quantities which comply with all ULS and SLS requirements, is reported by drawing a skull in the middle of the span for which this condition is identified. Moving the cursor over this skull, will inform the user on the actual criterion that can not be fulfilled with the specified cross-section dimensions and the maximum reinforcement ratio (for instance, limitation of concrete compressive stresses in SLS-QP)

Important note : PowerFrame provides the user with theoretical reinforcement quantities, which then need to be translated into a practical reinforcement design. During this translation of theoretical reinforcement into practical reinforcement, the user is urged not to re-use reinforcement bars at the upper or lower fiber for the right-hand or left-hand side of the beam cross-section. At any time and at each location, the sum of all practical reinforcement quantities at bottom, top, left- & right-hand side fiber needs to be equal at least to the sum of the theoretical reinforcement quantities as calculated by PowerFrame.

Calculated reinforcement quantities can be exported to the BuildSoft program ConCrete Plus, which allows to translate automatically the theoretical reinforcement quantities into a practical reinforcement plan and cutting list.

3.3.5 Modal analysis

PowerFrame allows the user to perform a modal analysis on the structural model defined so far. This model analysis will evaluate the structure's lowest N eigenfrequencies and corresponding eigenmodes. Launch the modal analysis through the menu entry 'Analysis – Modal analysis'.



PowerFrame asks the user to specify the number N of lowest eigenfrequencies to be calculated. To further refine the stiffness modeling of the structure during the modal analysis, all bars of the model can be subdivided into 10 segments. This will increase the accuracy of the eigenfrequencies and eigenvectors, but will also considerably increase the computational cost. The user should therefore only try to use this option in case of small to medium-size models. Anyway, it should be remembered that the calculation of the lowest global eigenmodes of the structures (lowest bending and torsion modes) does not normally require this refinement. The refinement does become important when higher modes are targeted, especially when highly localized modes are present.

Consistent with the static analysis capabilities, the modal analysis function does allow to consider or ignore the rigidity classification of the connections. This includes the possibility to consider hinges as near-hinges (see 3.3.1).

During the modal analysis, PowerFrame takes into account the effect of mass defined by the Selfweight load case. If the user also wants to consider the dynamic effect of masses which are related to permanent or quasi-permanent live loads, the user should be aware that dynamic masses can be assigned to nodes of the structural model (see 3.1.9.4.9).

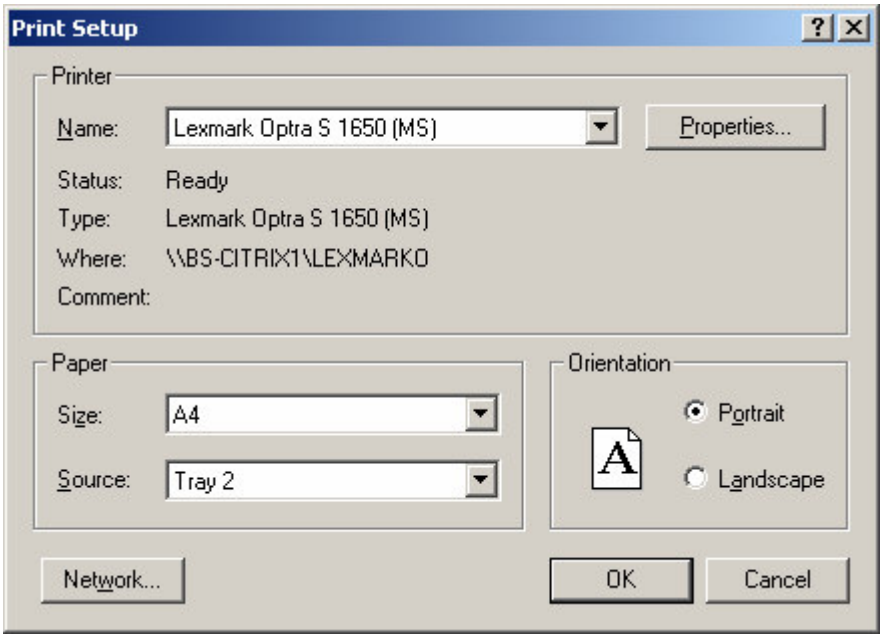
3.4 Printing model data and results

In this section of the PowerFrame reference manual, all aspects related to printing model data and results will be discussed:

- printer configuration
- printing a single window,
- printing an analysis report
- creating a comprehensive text format (RTF) file, which can be further processed by a word processor.

3.4.1 Printer configuration

In the main menu, the entry ‘File – Print setup’, allows the user to define the printer configuration through the dialogue window shown below.



This window corresponds to the MS Windows Print Setup dialogue and can be different depending on the actual MS Windows version the user is working with.

The user selects the requested printer, and if necessary, modifies the print parameters by using the button ‘Properties…’. In the lower half of the dialogue the user can define both paper size and orientation.

3.4.2 Printing a window

The contents of each of the 5 PowerFrame main windows ('Geometry', 'Loads', 'Plot', 'Data' and 'Results' can be printed.

For the first 3 types of windows, the actual contents will be rescaled automatically for maximum visibility on the selected paper format. During this rescaling operation, the height/width ratio of the window will be maintained. It should explicitly be noted that the rescaling applies to the actual window contents. In other words, if the user has previously zoomed in on a specific detail in the window, the Print Window function will only print the detail view.

For both tabular type of windows, the complete tables are sent to the printer. The scroll position of the table inside the window does not affect this at all.

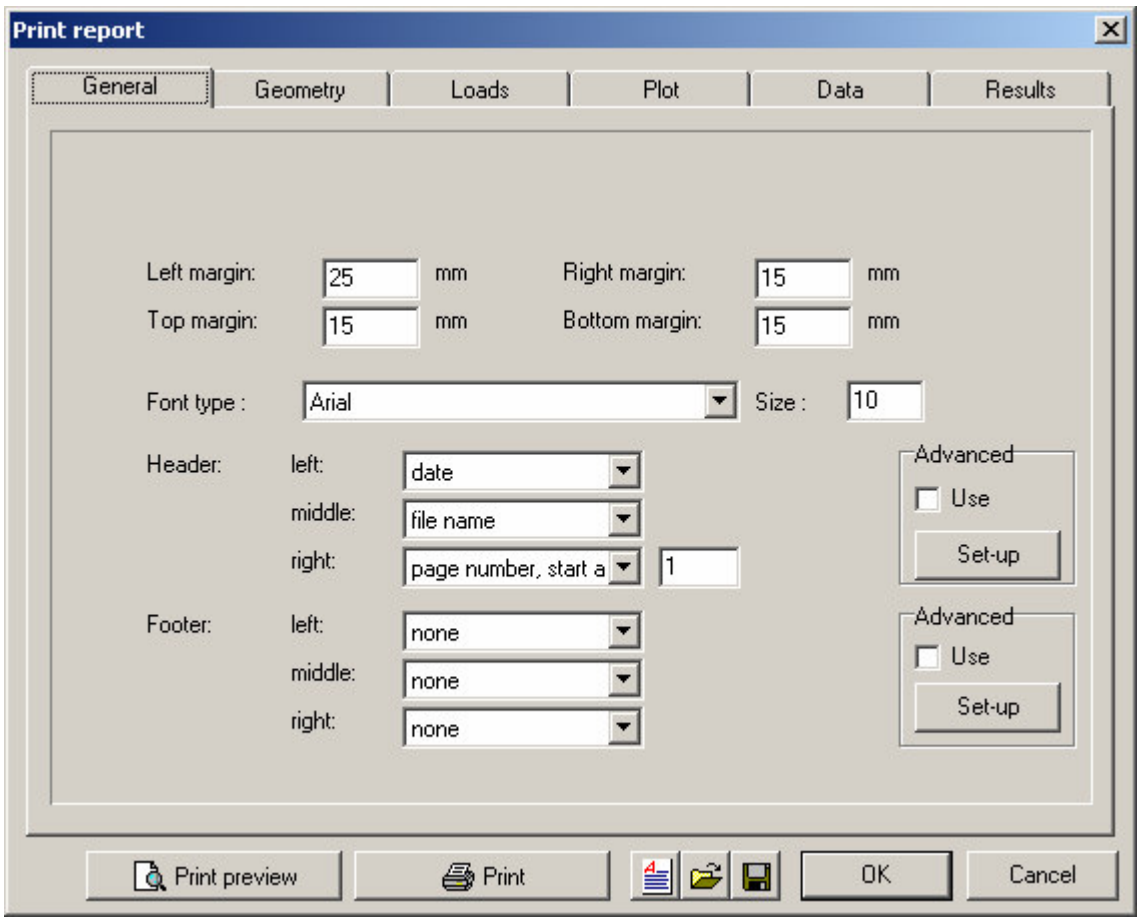
To print the contents of a specific window, the user should verify that the window he wants to print is the active window. If this is the case, 3 possibilities exist to actually print the window :

- use the main menu entry 'File – Print window' ;
- use the key combination CTRL + P on user's keyboard ;
- use the icon in the main icon bar.

3.4.3 Printing a report

To print a report, use the main menu entry 'File – Print report…'. A dialogue window appears which contains 6 tab pages. The first tab page allows to specify the general print parameters, whereas each of the following tab pages corresponds to one of the PowerFrame main windows.

3.4.3.1 Tab page 'General'

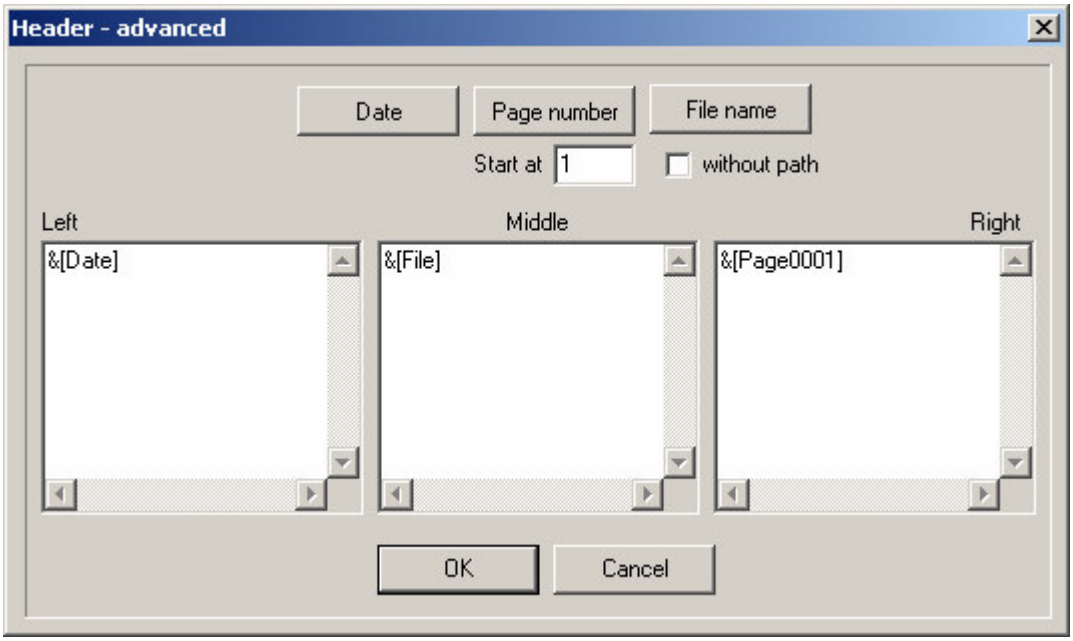


First of all, the user can define the left, right, top and bottom margins which should be left blank by the printing process.

Next, the user can specify header and footer for each page of the report. Both header and footer contain 3 areas (left, middle and right). For each area, he can use a pull-down menu to define its contents:

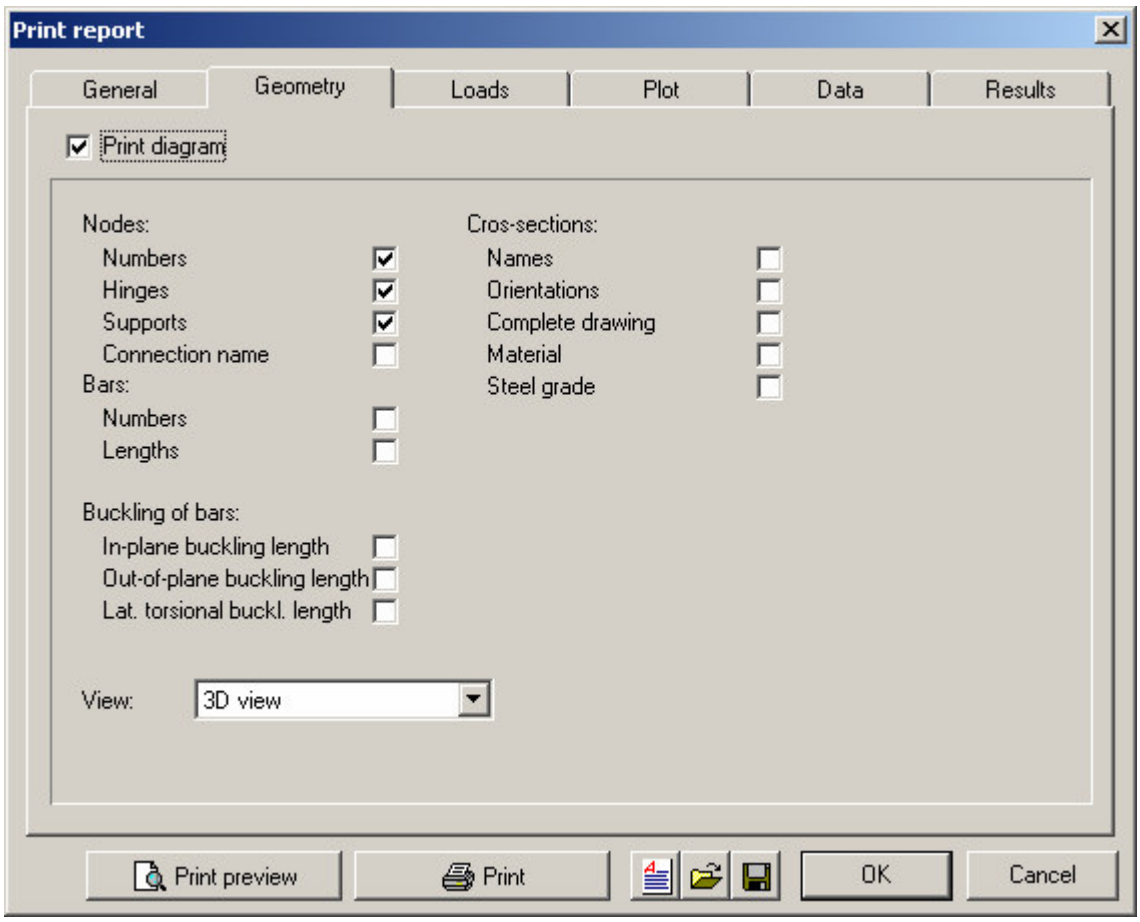
- empty
- date (print date)
- project name (name of the PowerFrame file, including the complete path definition)
- page number (starting from a number defined by the user. The first page which is printed will bear the start number specified by the user.)
- a text which can be freely specified by the user.

For a more advanced definition of headers and footers, use the advanced setup buttons in the above window dialogue. This will schedule a new dialogue (shown below), which allows to specify the content of the 3 zones (left, middle & right).



To introduce information in one of the areas, position the mouse in the relevant zone of the above dialogue. Now define for instance the customized text, possibly spread out over several lines. To introduce data, page number or file name, just click any of the buttons on top of the dialogue, making sure you have selected the appropriate zone in the dialogue first.

3.4.3.2 Tab page ‘Geometry’



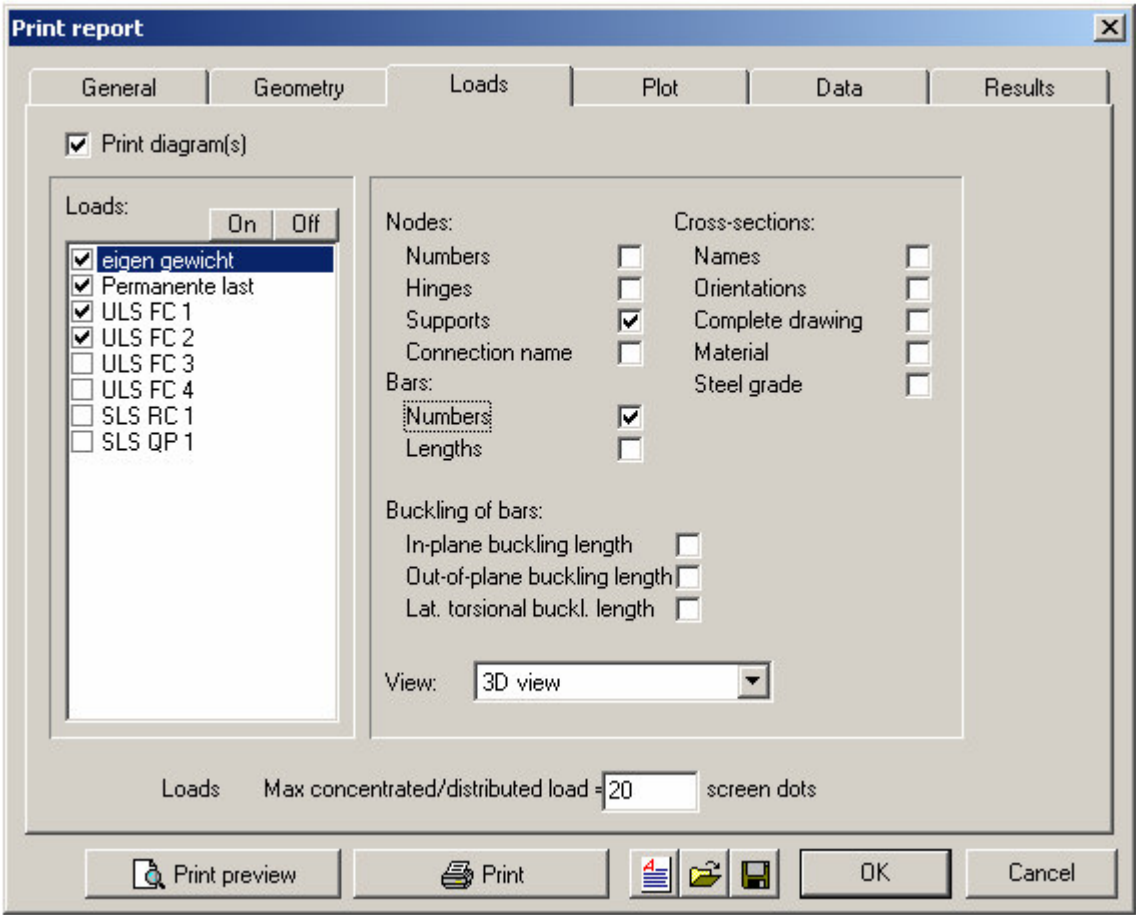
First, the user should select the option ‘Print diagram’ to be able to include geometry information in the report. Then, he can specify the data which are to be included in the report. This is very similar to the specification of the information shown in the ‘Geometry’ window (see 3.1.1), but it is important to realize that both definitions are made completely independent from one another.

On the tab page, the user can also specify the viewpoint to be used on the print-out. This viewpoint can be different from the viewpoint that is actually in use in the ‘Geometry’ window itself. If the user selects a 3D view, the same perspective will be used in the report as in the actual window. However, the visible part of the model will always be resized for maximum visibility on the selected paper format, independent of the zoom factor in the ‘Geometry’ window.

3.4.3.3 Tab page ‘Loads’

Similar to the tab page ‘Geometry’, the user first needs to specify whether he actually wants to print loads information in the report. If this option has been activated, you can select in the left-hand column which load cases and/or load combinations are to be included in the report. For each selected case or combination, a drawing will be generated. Note that the buttons ‘On’ and ‘Off’

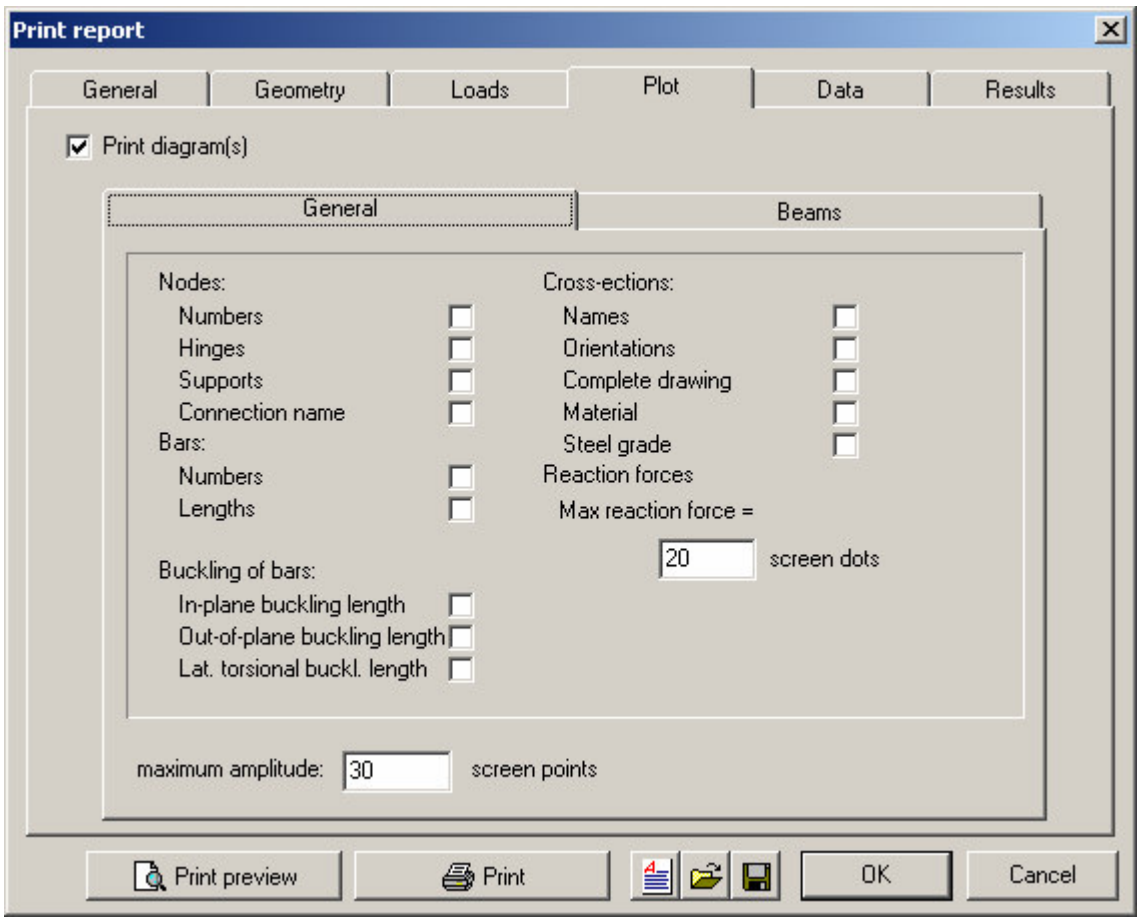
at the top of this list allow to select / deselect all load cases & combinations simultaneously. On the right-hand side of the tab page, the same parameters as found on the 'Geometry' tab page can be found. The user will note that the definitions made on the 'Loads' tab page are completely independent of the general visualization parameters defined for the 'Loads' window (see 3.1.1).



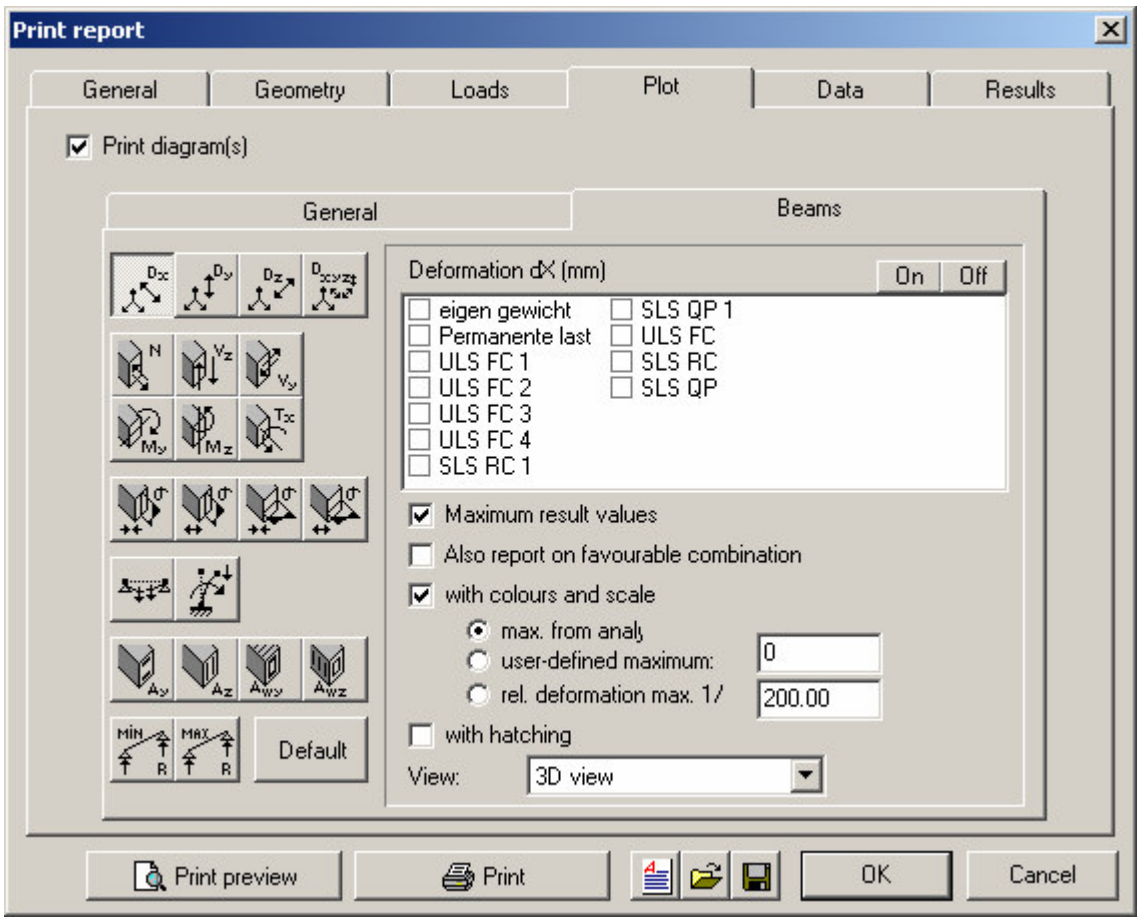
3.4.3.4 Tab page 'Plot'

First, the user will specify he wants to include plot data in the analysis report by selecting 'Print diagrams' in the tab page. He will then get access to 2 additional tab pages "General" and "Beams".

On the first tab page 'General', the user can again specify (similar as for the tab pages that were discussed previously) which general information is to be included with each plot diagram. At the bottom of the tab page, the user can specify the number of screen points to be used for the representation of the maximum deviation relative to the undeformed structural members. Note that all parameters are completely unrelated to the ones defined directly on the 'Plot'-window (see 3.1.1).



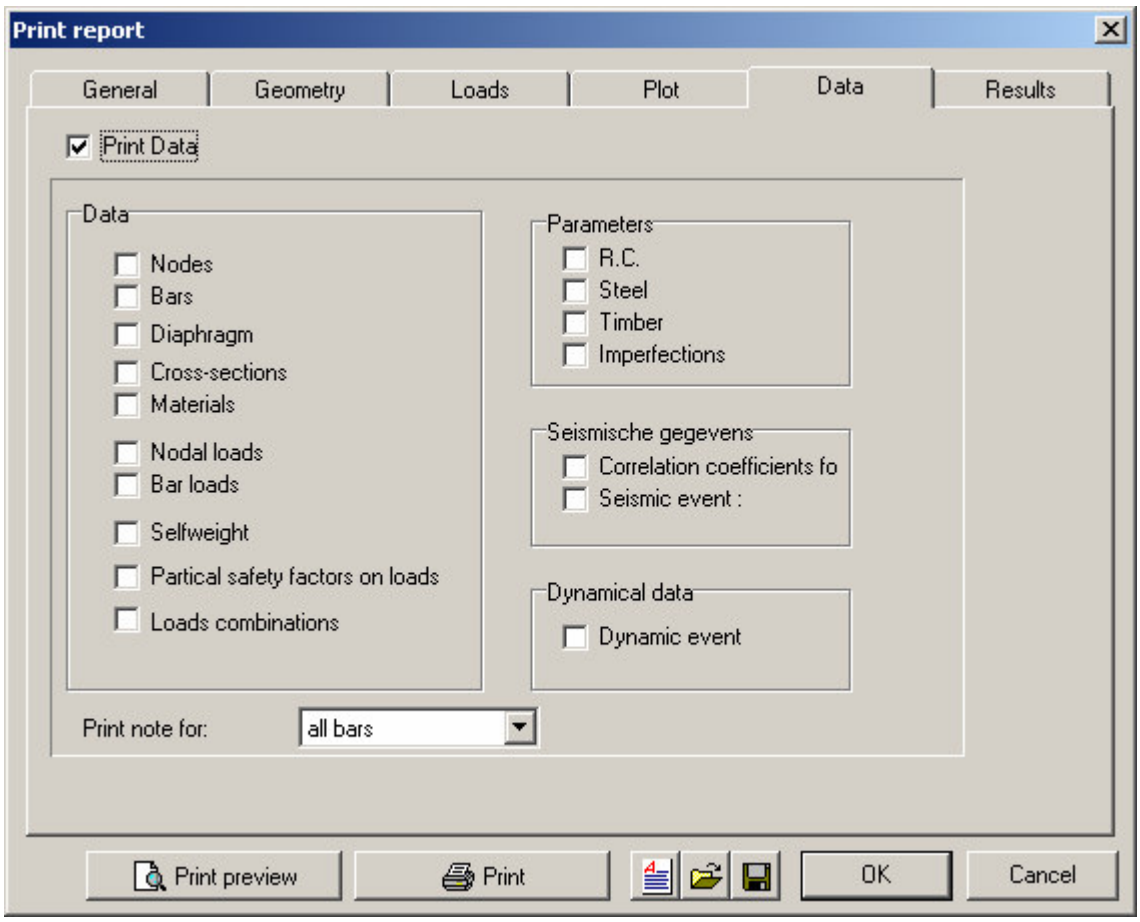
The second tab page ‘Beams’ contains all parameters that are needed to define the actual contents of the different plots to be made. First, the user will select one of the icons on the left-hand side corresponding to the results type that needs to be reported. Then, he will refine the definition by selecting the load case and/or combination for which the active results type needs to be included in the report.



3.4.3.5 Tab page ‘Data’

The tab page ‘Data’ allows to print **tabular** data concerning cross-sections, material properties, loads, ... Having selected the ‘Print data’ option, the user can further refine the specification of the actual data from the ‘Data’-window to be included in the report.

Important note: the data presented in the ‘Data’-window is limited to the visible parts of the model only. In the tab page ‘Data’, the user can specify whether he wants to print tabular data for the complete model, or for the visible bars only.



If the option 'Cross-sections' is selected, all cross-section data and properties are printed in the format used by the dialogue window for the definition of cross-section properties based on cross-section types (see 3.1.8.3.15). If the option 'Materials' is selected, the properties of the materials that are actually used in the PowerFrame project are printed in the format used by the dialogue window for the definition of new material properties (see 3.1.6).

If the option 'Load combinations' is selected, the applicable safety and combination factors as used for all load combinations, will be included in the report. The options in the right-hand part of the dialogue allow to include in the report those parameters that are actually used for the design checks (R.C., steel or timber characteristics and description of global imperfections).

3.4.3.6 Tab page 'Results'

The last tab page allows to print analysis results in a tabular format. A distinction is made between global analysis results (results at member ends) and detailed analysis results (results at 11 points along each bar). Those result types correspond to the contents of the 'Results' window as follows:

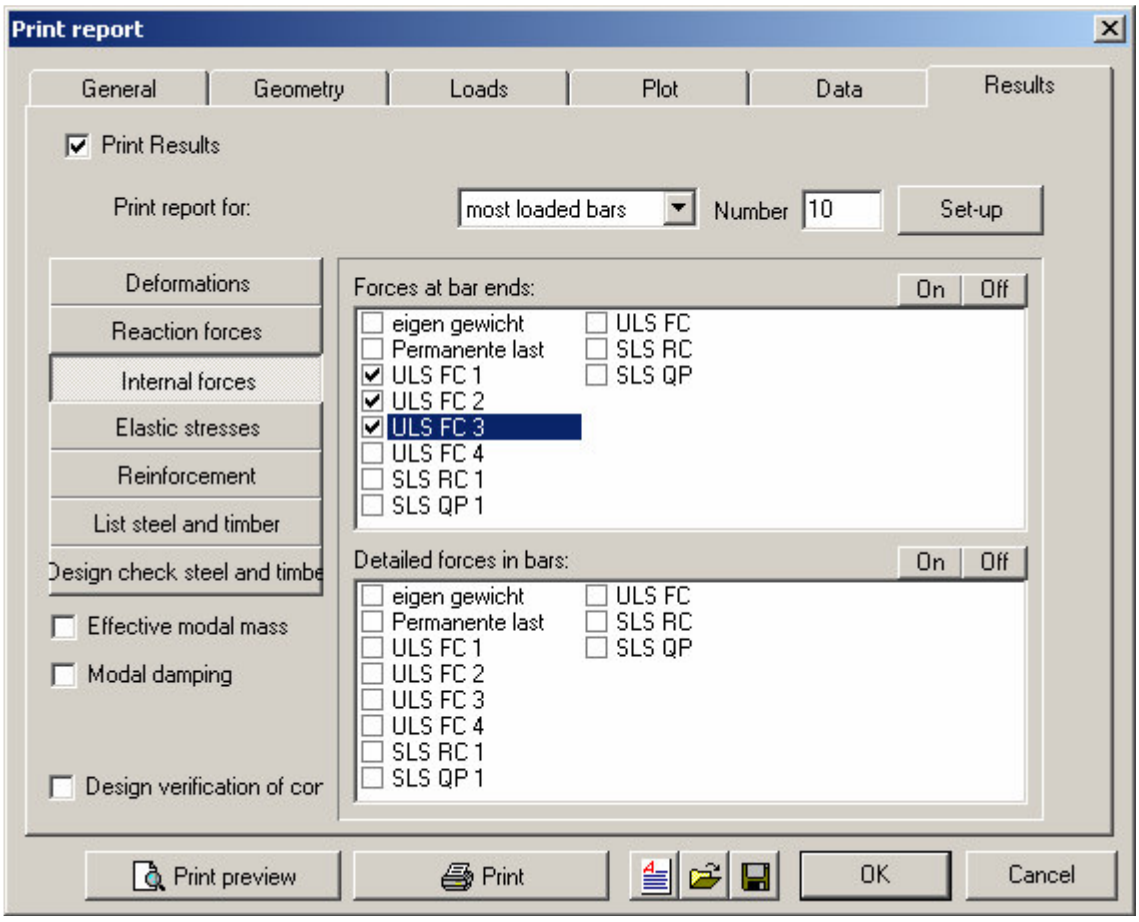
- Global results are shown in the 'Results'-window in case no bars or more than one bar are selected in the 'Plot'-window

- Detailed results are shown in the ‘Results’-window in case only one bar is selected in the ‘Plot’-window

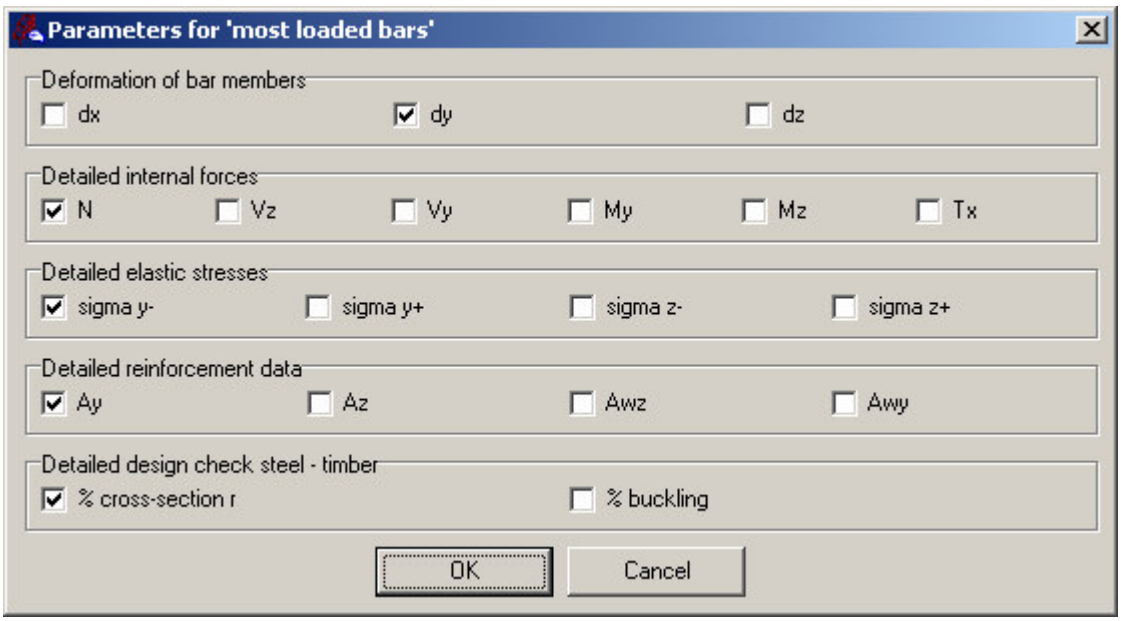
The user first activates the tab page by selecting the option ‘Print results ‘. Through the pull-down menu, he can then specify for which bars analysis results need to be printed:

- all bars ;
- all visible bars ;
- most loaded bars only.

By using the option ‘All visible bars’, the user has direct control over the bars for which he prints the analysis results. To do so, select the bars of interest in the ‘Plot’-window and then make all other bars invisible.



If the user requests analysis results to be printed for the most loaded bars only, he further needs to specify the number of members that must be included in the report. Furthermore, the button ‘Set-up’ allows to define the criteria that are used to specify the actual meaning of the criterion ‘most loaded’.



Suppose the user has asked the 10 most loaded bars to be included in the analysis report and that he has further selected axial force N and bending moments M_y & M_z to be considered as selection criteria. This will include in the report

- 10 bars with maximum axial force
- 10 bars with maximum value of My
- 10 bars with maximum value of Mz

Accordingly, the total number of bars included in the report will be 10 at a minimum (in case the above described sets of 10 bars are identical) and 30 at a maximum (in case the above described sets of 10 bars are completely different).

To further complete the definition of the results to be printed to the report, use the buttons at the left-hand side of the dialogue to select 'Deformations', 'Reactions', 'Internal forces', ... At the right-hand side of the dialogue, the user can then further specify the load cases and/or combinations for which he wants to print global analysis results and/or detailed analysis results.

Some further clarifications with respect to the different results types that can be selected in the dialogue window:

- the button 'Deformations' allows to print deformations at nodes (or at intermediate points) in tabular format.
- the button 'Reaction forces' allows to print reaction forces at external supports in tabular format.
- the button 'Internal forces' allows to include following internal forces in a tabular report : axial force N, shear forces $V_{y'}$ and $V_{z'}$, bending moments $M_{y'}$ and $M_{z'}$ and torsion moment T_x.

- the button ‘Elastic stresses’ gives access to elastic stresses due to axial forces and bending moments for homogenuous elastic material (not available for materials of type concrete).
- the button ‘Reinforcement’ will present calculated longitudinal and transverse reinforcement quantities in a tabular format.
- the button ‘List steel and timber’ relates to the results of cross-section strength and buckling stability checks (expressed as % of maximum strength).
- using the button ‘Design check steel and timber’, the user can also include the detailed results of cross-section strength and buckling stability verifications.

Finally, the option ‘Verification of connections’ allows to include the results of the verification analysis on steel connections that were previously designed with PowerFrame Master (see 4.3), as part of the analysis report.

3.4.3.7 Additional options

3.4.3.7.1 Saving and reading printing preferences

In the previous sections, it was exposed how the analysis report can be tailored to the user’s specific demands. Having gone through all necessary steps, the user may want to re-use the results of his specification work with other PowerFrame projects as well. To do so, the user should save the printing preferences he has defined using the icon at the bottom of the dialogue window. It will then be possible to load those printing preferences in another PowerFrame project, using the icon in the dialogue window. Of course, there is no guarantee that the number of load cases and load combinations will be the same in both projects. Therefore, the load cases and load combinations selected in the tab pages ‘Plot’ and ‘Results’ are not saved in the preference file, but the selected envelopes (ULS, SLS RC and SLS QP) are saved and can be re-used.

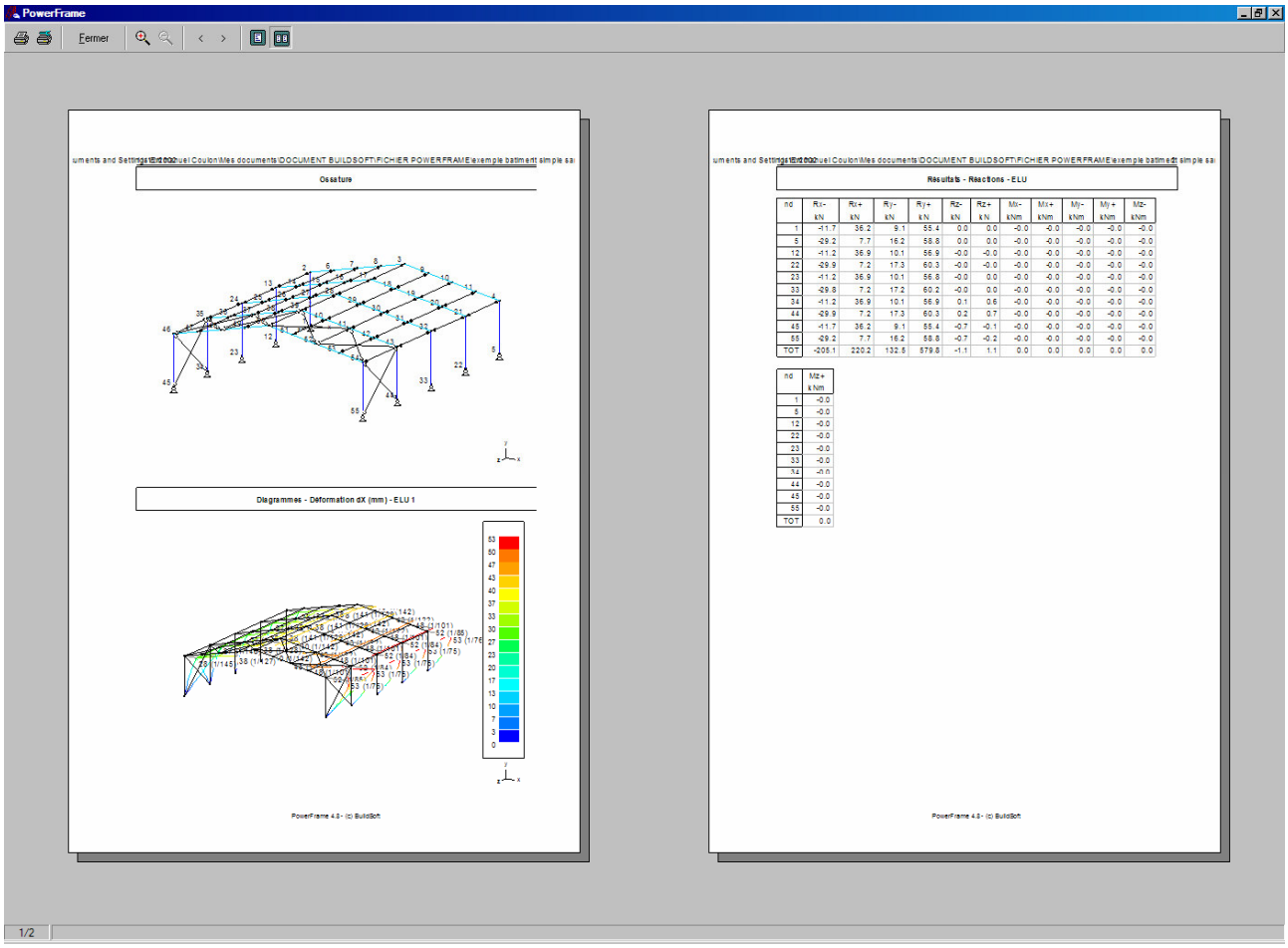
3.4.3.7.2 Saving reports as RTF file

Once the definition of the printing preferences has been completed, the analysis report can be printed on paper. Alternatively, the report can also be written to a RTF (Rich Text Format) file. This file can be used with most word processors, giving the user the possibility to further edit and complete the document (for instance, include the company logo) and thus allowing for a full customization of the PowerFrame reports.

To actually save the report to RTF, use the icon at the bottom of the dialogue window.

3.4.4 Print preview

Before actually printing the analysis report to paper, the user can preview it and check whether it really meets his expectations through the icon or through the main menu entry ‘File – Print Preview’.



The user will now get a print preview window on the screen, similar to the one shown above. The first 2 icons and allow to launch the print job and to define/modify the printer setup.

Using the magnifying glass , a rectangle can be drawn on the page preview to zoom in on the selected area. To return to the original view, use .

Finally, a number of icons allow the user to easily explore the complete preview document :

- and allow for quick navigation, providing shortcuts to the next and previous page ;
- and allow to show 1 or 2 pages in the preview window.

To complete the preview process, press the 'Close'-button.

3.5 Saving and opening projects

3.5.1 Saving a PowerFrame project

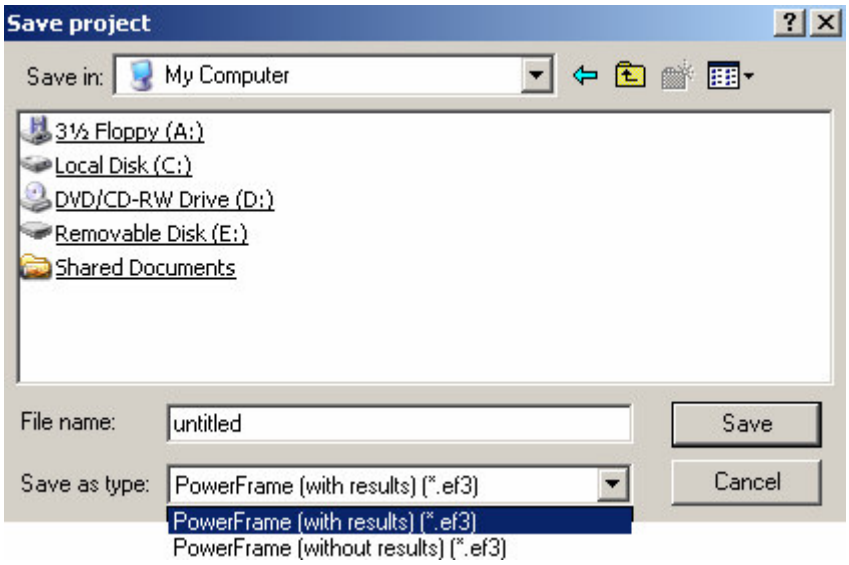
To save a PowerFrame project, the user should utilize the menu entry 'File – Save ' or the icon. Alternatively, the menu entry 'File – Save as…' can also be used.

PowerFrame projects are saved on the computer's hard disc with file extension '.ef3'. The difference between the 'Save' and 'Save as…' menu entries can now be described as follows:

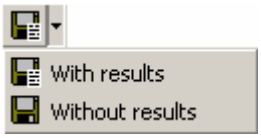
- if the user has already saved his PowerFrame project previously, 'Save' will save an updated version of the project to the same .ef3 file, now including also the changes introduced into the PowerFrame project since the last 'Save'-operation. At the same time, the extension of the previously saved version of the PowerFrame project will now be changed into '.ef!', which creates a back-up of the project.
- if the user has already saved his PowerFrame project previously, 'Save as…' will save the project in a new file. Thus, the user can for instance write different "versions" of the analysis project to different physical files on the hard disc.

PowerFrame also allows the user to save projects without the analysis results. Several possibilities exist to do this:

- through the menu entry 'Edit – Preferences', a dialogue window is scheduled (also refer to 3.6) in which the option 'Results are stored when saving the project' can be unselected. Consequently, all Save operations will only save model data to the project file, but no analysis results
- in the 'Save project' dialogue window, a pull-down menu offers the user the choice to save the project with or without the analysis results.



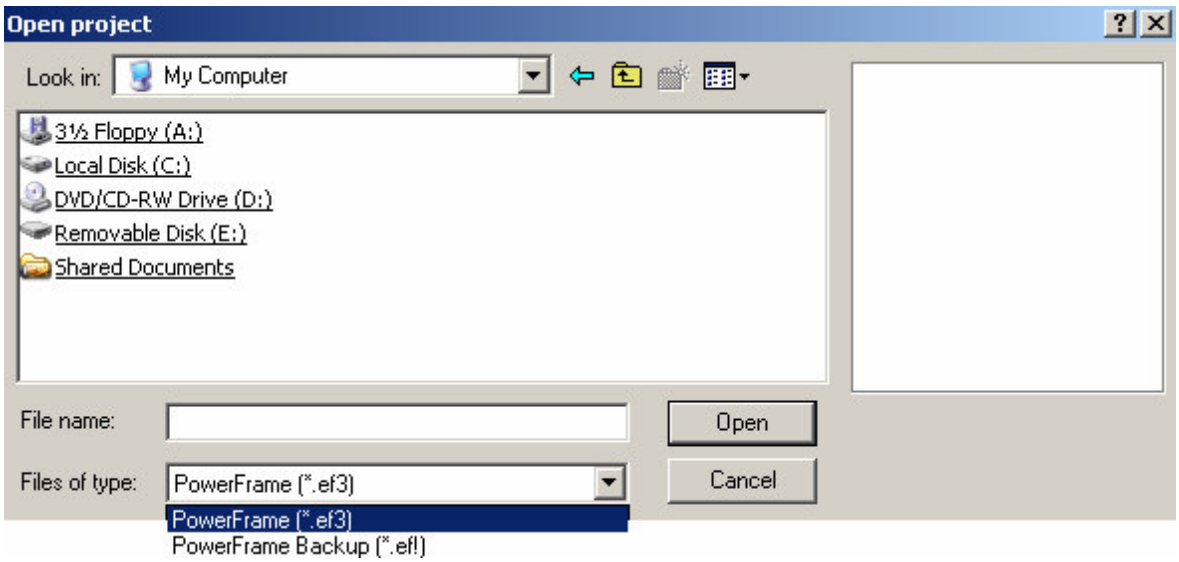
- when the icon is used to save a PowerFrame project, the pull-down arrow allows the user to specify how the project should be saved (see below).



3.5.2 Opening a PowerFrame project

Next to the 'standard' PowerFrame projects (with extension '.ef3'), PowerFrame can also directly open back-up projects (with extension '.ef !'). To do so, use the menu entry 'File – Open…' or directly use the icon . It should be noted that the pull-down part of this icon allows the user to open directly the most recently used PowerFrame projects. The list of the most recently used projects will automatically appear in a pull-down menu when the arrow is pressed down.

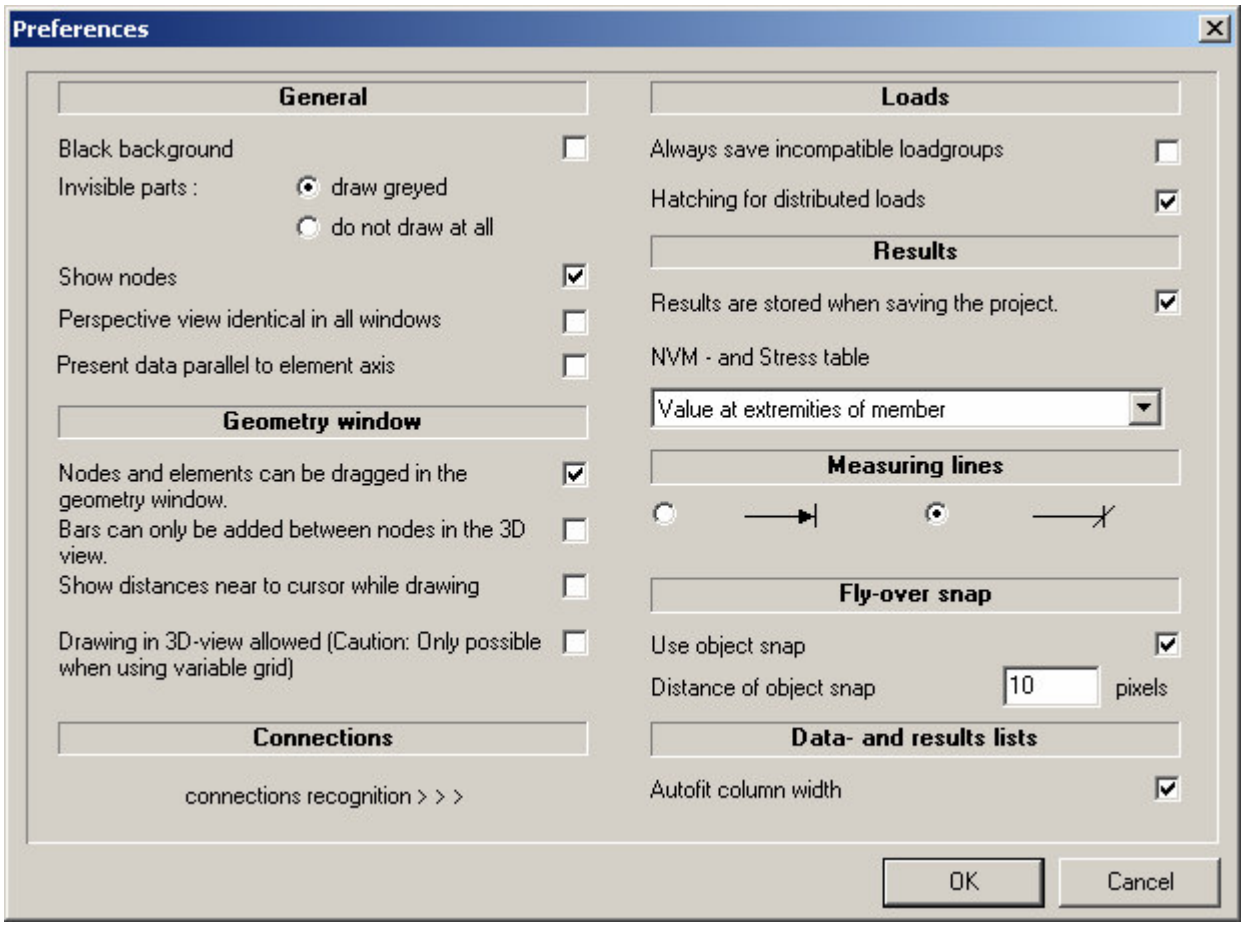
In case a project is opened using the complete 'File – Open..' dialogue, a pull-down menu allows to specify the type of file to be opened ('.ef3' or '.ef!').



3.6 Preferences

3.6.1 General parameters

The menu entry ‘Edit – Preferences’ gives access to a dialogue, in which a number of global preferences can be defined, related to different aspects of the work with PowerFrame.



General:

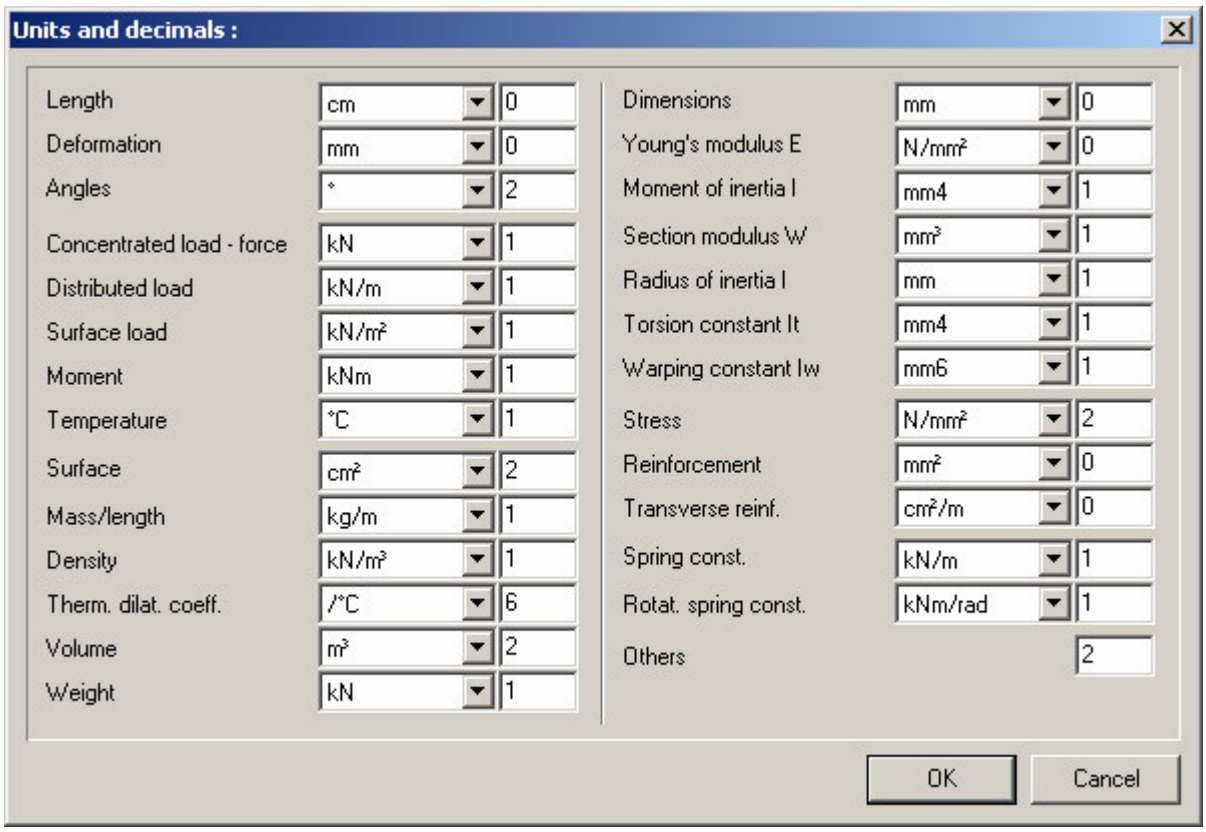
- selecting the option ‘Back blackground’ will activate a black background in the ‘Geometry’, ‘Loads’ and ‘Plot’ windows ;
- invisible bars can either be grayed out on the screen (which is usefull to understand how the visible bars related to the remainder of the structural model) or completely be omitted from the graphical visualization (which usually makes the presentation of the results a lot easier to understand) ;
- nodes of the analysis model are usually highlighted by means of a small rectangle. Optionally, this rectangle can be omitted ;
- in case a 3D view is selected in the graphical windows, the viewpoint and perspective can be different in each window. However, it is usually more convenient to have the 3 graphical windows aligned in terms of viewpoint & perspective.

Whereas the options specifically related to the ‘Geometry’-window are fairly self-explaining, some additional explanation is required with respect to the ‘Loads’ option. During the definition of load cases, specific load groups can be declared as mutually exclusive or incompatible. This option will make sure that the definition of loads incompatibility is saved for later re-use.

The ‘Fly-over snap’ options allow to control PowerFrame’s intelligent cursor. It can be switched on or off, while the snap resolution can also be specified.

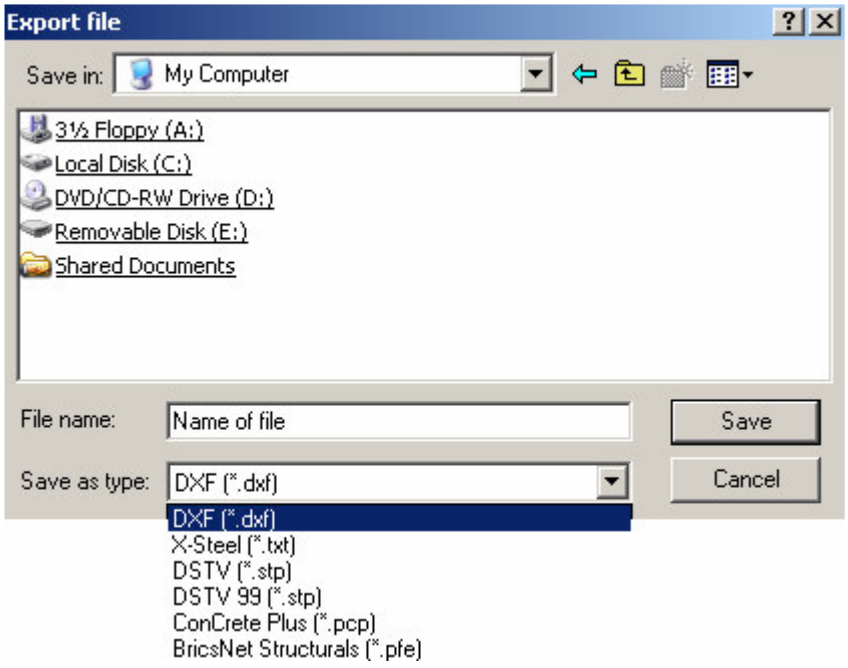
3.6.2 Units and decimals

Use the menu entry ‘Screen – Units and decimals…’ to specify in which units you want to enter model data and you want to display analysis results. The level of precision can be adjusted by modifying the requested number of decimals.



3.7 Importing and exporting data

To import model data from external software programs for re-use within PowerFrame or to export model data that the user has created with PowerFrame towards an external software program, use the main menu entries ‘File – Import…’ and ‘File – Export…’. The dialogue window that appears requires the specification of a file format to be used for data exchange with applications external to PowerFrame.



3.7.1 Import/export to DXF

The DXF format is supported by most CAD programs for the exchange of drawing information. In the context of PowerFrame, the information that is read from or written to DXF relates to model geometry: co-ordinates of nodes, connection of nodes by bars, …. It does not include the attributes of nodes and bars, like eg. definition of boundary conditions, cross-section properties, material characteristics, …

CAD-programs usually organize data in a number of layers. During the import of data from DXF, PowerFrame allows the user to limit the import process to specific layers created by the CAD program. During import, PowerFrame will automatically translate each layer into a different type of element (see 3.1.8.7). At the time of export, the different element types are translated into individual layers written to DXF. This mechanism can be used to control how and what data should be exchanged between PowerFrame and the external CAD program.

3.7.2 Import/export to DSTV

DSTV-files, also referred to as STP- or STEP-files, not only contain geometry information (as DXF files do), but can also include attributes associated to nodes and lines of the model geometry. DSTV-files are used by most professional steel modelling software programs as input for the manufacturing process based on a 3D drawing model.

3.7.3 Export to ConCrete Plus

ConCrete Plus is a software program developed by BuildSoft, which enables engineers to translate theoretical reinforcement quantities calculated by PowerFrame, into a practical reinforcement design (including reinforcement drawings and bar charts).

To transfer PowerFrame elements to ConCrete Plus, the user should first check whether the ‘Plot’-window is activated, and select one of the 4 possible reinforcement results to be displayed. He should then select one or more elements for which he wants to transfer theoretical reinforcement quantities towards ConCrete Plus, then go to the main men entry ‘File – Expor…’ and choose the file format ‘ConCrete Plus (*.pcp)’. In case reinforcement data of

a single elements needs to be transferred to ConCrete Plus, it is also possible to directly Copy/Paste the data between the 2 applications.

The word ‘element’ in the above paragraphs refers to a number of adjacent bars along a single line. One such element is imported in ConCrete Plus as a single (multi-span) beam.

3.7.4 Export to Microsoft Excel

PowerFrame enables you to export data tables and result tables to an external spreadsheet program. Activate the appropriate window and choose the instruction ‘Edit – Copy’. Next, open your spreadsheet and use the ‘Edit – Paste’ function. The table now appears in your worksheet and it can be further processed.

4 Connection design

The PowerFrame Master version not only allows to perform a global design analysis of a structural frame, but also provides the possibility to further use the global design results for a detailed design of all steel connections that are part of the frame. This is done using the technology of the BuildSoft product PowerConnect, integrated within the PowerFrame environment using the DLL (Dynamic Link Library) technology.

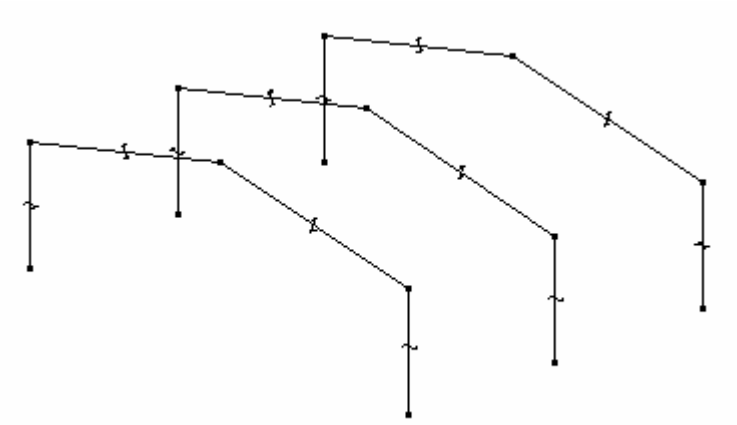
Thanks to the interaction between the global frame design analysis and the detailed connection design analysis, the internal forces calculated by PowerFrame can directly be used as loads on the connection to be designed, rather than requiring a manual, time-consuming definition. As the interaction is a bi-directional one, a further advantage results from the fact that the connection stiffness calculated by the PowerConnect technology can be introduced in the global design analysis of the frame, allowing for a more economic and optimal design.

4.1 Detail design of connections

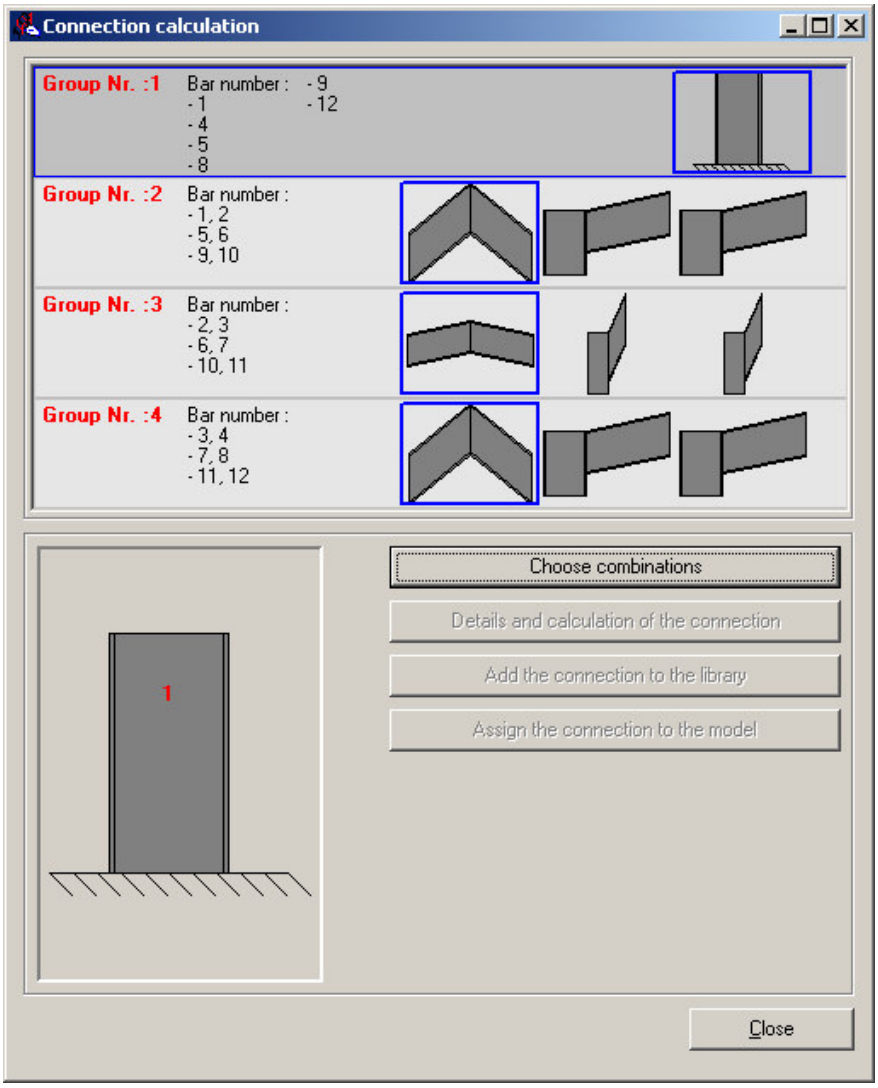
Before actually starting with the detail design of any connection in the structural frame model, it is advised to complete the cross-section resistance and member buckling stability verifications. This ensures that all cross-sections have been assigned realistic properties for the given type of structure and loading. In any case, it is absolutely essential to have the results of a global elastic analysis available, and to actually have loads data which can be passed on to the detail calculation of the connection.

To start the connection detailing, select those bars which are connected by all connections to be designed. Then use the menu entry ‘Analysis – Link to PowerConnect’ or use the icon. A dialogue window will appear presenting an overview with all connections that can be handled by PowerConnect. Besides, all connections with identical configuration will be grouped into one connection model. In this way, large groups of connections can be detailed in no time.

An example will illustrate this operating procedure. Let’s have a look to the structure below, for which we want to detail all connections. All bars are made invisible except for the three structural frames. After selecting the whole structure, use the menu entry ‘Analysis – Link to PowerConnect’ or use the icon, to start the connection design analysis.



In case PowerConnect can actually handle one or more connections that you selected, a dialogue window is presented to start the detailing process. If PowerConnect for some reason cannot continue this process based on your selection, a warning will be issued.



PowerFrame schematically presents the type of connection to be designed, along with the numbers of the nodes and bars involved. In this example, a

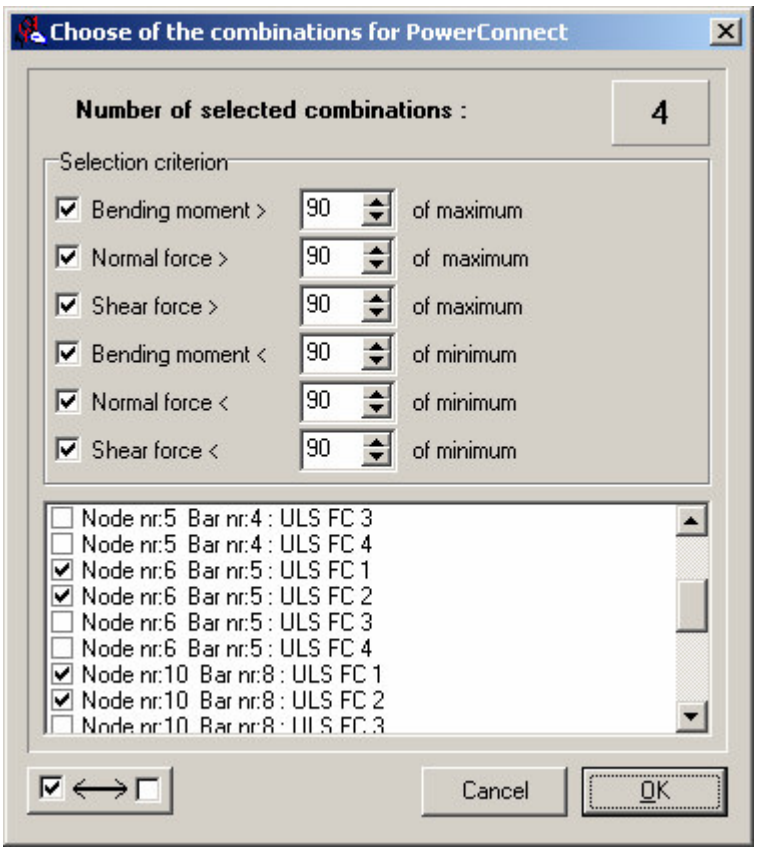
column base is recognized for bar numbers 1, 4, 5, 8, 9 and 12. They are joined in one group calling ‘Group Nr.1’. In this way, only one column base is defined and detailed, subject to all possible (combinations of) internal forces under consideration. This column base can then be assigned to all relevant nodes.

Another type of connection is found under ‘Group Nr.2’. For these three pairs of members, you‘ll still have to define the connection topology. In case of a column-beam connection you also have to specify which member is to be considered as the primary bar and which is to be considered as secondary.
The same logic will be encountered for other connection types.

First step:

After all connections have been recognized, they can be detailed one after the other. However, before actually starting the connection detailing, you‘ll have yet to choose the load combinations which are to be used for the connection design. It is obvious that not necessarily all load combinations that were considered during the global PowerFrame analysis are critical for the connection calculation. Therefore, the link between the global frame analysis and the detailed connection analysis allows to filter the load cases used by PowerFrame, based on % of the maximum internal forces calculated by PowerFrame. The filter threshold can be specified independently for the different types of internal forces (axial force, shear force, bending moment). The user should be careful not to define the threshold values too high. For instance, if all filter values are defined as 95%, the risk exists that those combinations in which M, N and V reach a value of 90% simultaneously are not included in the selection, while the combination with >95% values for M but fairly low values for V and N will be included. Nevertheless, the first type of combination will usually be more critical, because of the “simultaneous peaking” of all internal forces.

As an example, select the first group in the list and use the button ‘Choose combinations’.



In the above dialogue, it can also be seen that filter values can be defined separately for negative and positive values of the internal forces. At any time, the number of selected combinations is indicated while the selection can always be extended or limited by manually selecting or unselecting combinations in the list.

Note : as the connection design analysis is always based on the evaluation of ultimate limit states (ULS), only the ULS combinations are presented in the above dialogue.

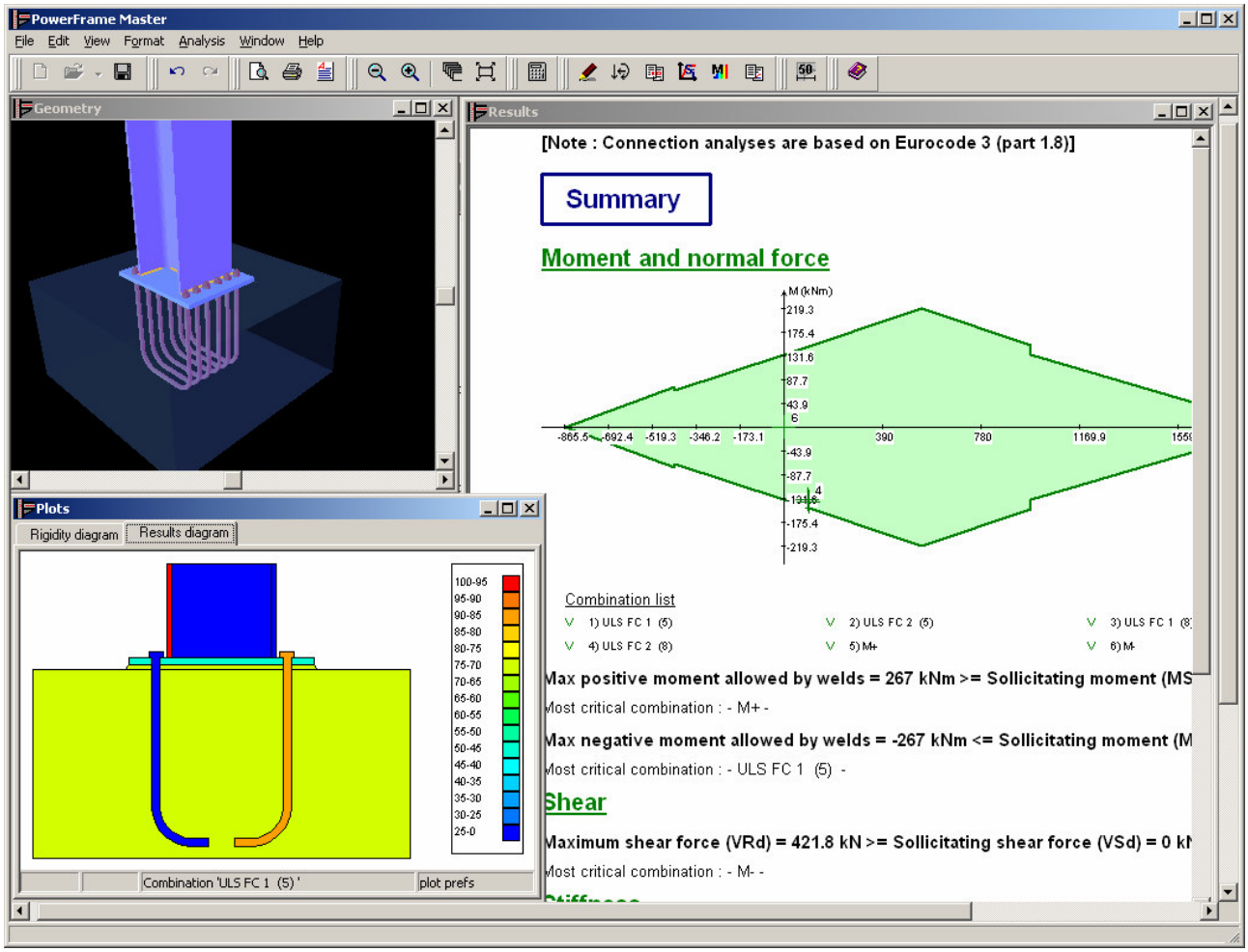
Once the appropriate combinations have been selected, the user should confirm the choice by the 'OK' button, and then return to the previous dialogue window. The 'V' sign in front of the button 'Choose combinations' reveals that this step is finished. You are now ready to use the button 'Details and calculation of the connection'.

Note : If the user does not have a PowerFrame Master license but wishes to transmit the previously generated data to a colleague who does have this type of license, he can use the *icon to save the data to an external file. This file can then be further used with a PowerFrame Master license.*

Second step:

Using the button ‘Details and calculation of the connection’ now brings you in the PowerConnect environment where you can proceed with the detailed connection design. If the selected node is detailed for the first time, a further dialogue window asks you to specify the type of connection to be used for the design.

Once you are truly in the PowerConnect environment, you should refer to the PowerConnect reference manual for further information.



When you have completed the detailed connection design with PowerConnect, you should switch back to the PowerFrame global analysis through the main menu entry ‘File – Quit’. At that time, a second ‘V’ sign appears in front of the button ‘Details and calculation of the connection’. The results are now saved, even when you switch over to another type of connection.

In case the stiffness has been calculated, PowerFrame asks which stiffness value must be further used in the global model, either for a positive or a negative bending moment. Indeed PowerConnect has calculated both stiffness values (in case combinations where used that contained positive and

negative bending moments), but only one stiffness value can be accounted for during the further global frame analyses.

The values presented depend on the ratio of the sollicitating moment M_{Sd} and the resisting moment M_{Rd} of the connection. In case of positive bending moments, the following rule is used to determine the connection stiffness for transfer to PowerFrame:

- if $M_{Sd} < 2/3\ M_{Rd} \rightarrow S_{j,ini}$
- if $M_{Sd} > 2/3\ M_{Rd} \rightarrow S_j$

In case of a double connection (beam – column – beam), the connection stiffness needs to be specified both at the right- and left-hand side of the connection.

Third step:

Once the connection has been detailed, it can be saved in a so-called connection library by clicking on the third button 'Add the connection to the library'. Don't forget to first specify a name for the connection that is transferred. Again, a 'V' sign affirms your operation.

Note: this operation is necessary when you want to assign this connection to nodes.

Fourth step:

The last step consists in introducing the connection properties into the PowerFrame global model. In case a stiffness has been calculated, those stiffness value will be assigned to the ends of the bars that meet at the node.

To do so, click on the button 'Assign the connection to the model'. The original name 'Group nr.1' at the top of the window is now replaced by the new name.

Since the model properties are changed, the results (internal forces) are no longer coherent to this new model. However, it is still allowed to detail other connection types without re-running the global analysis, at least as long as the current dialogue window hasn't been closed.

All steps have now been discussed. The same procedure can be used for all other connection types.

Upon completion of the above described transfer process, the connection is stored in a connection library accessible directly from within the PowerFrame environment.

In case a connection is already present in this connection library, PowerFrame will ask the user to confirm if the new connection properties should also be assigned to all nodes that are associated to the connection.

4.2 Connection library

The connection library can be accessed through the icon. First of all, it is necessary to make a distinction between

- ✓ an internal library, associated with the project itself,
- ✓ an external library which has no relationship to any particular project but is generally available to all projects.

At the moment a new connection is assigned to a node in the active model, this connection will be stored only in the internal connection library. In this way, the connection is saved along with the Powerframe model so that any other PowerFrame Master user can ask for this information when the project is opened.

On the other hand, the external library is called 'ConnectionLibraryPF.clf' and is located under the installation directory of PowerFrame. PowerFrame Master users have access to this library from within any PowerFrame project.

At the start of a project, no connections are stored in the internal library yet. The user will, during the design process, add connections to this library. Those connections can then be assigned to specific nodes of the PowerFrame model. Nodes are selected by selecting the adjacent bars, only one node should be selected at a time.

Depending on the selection of bars, several scenarios are possible when the icon is used:

- case 1: a connection has already been assigned to the selected node. In this case, it will be presented by the program when the icon is used.

- case 2: no connection has been assigned yet to the selected node. However, one or more connections are available in the library which fit to the selected node. They will be presented by the program when the icon is used.

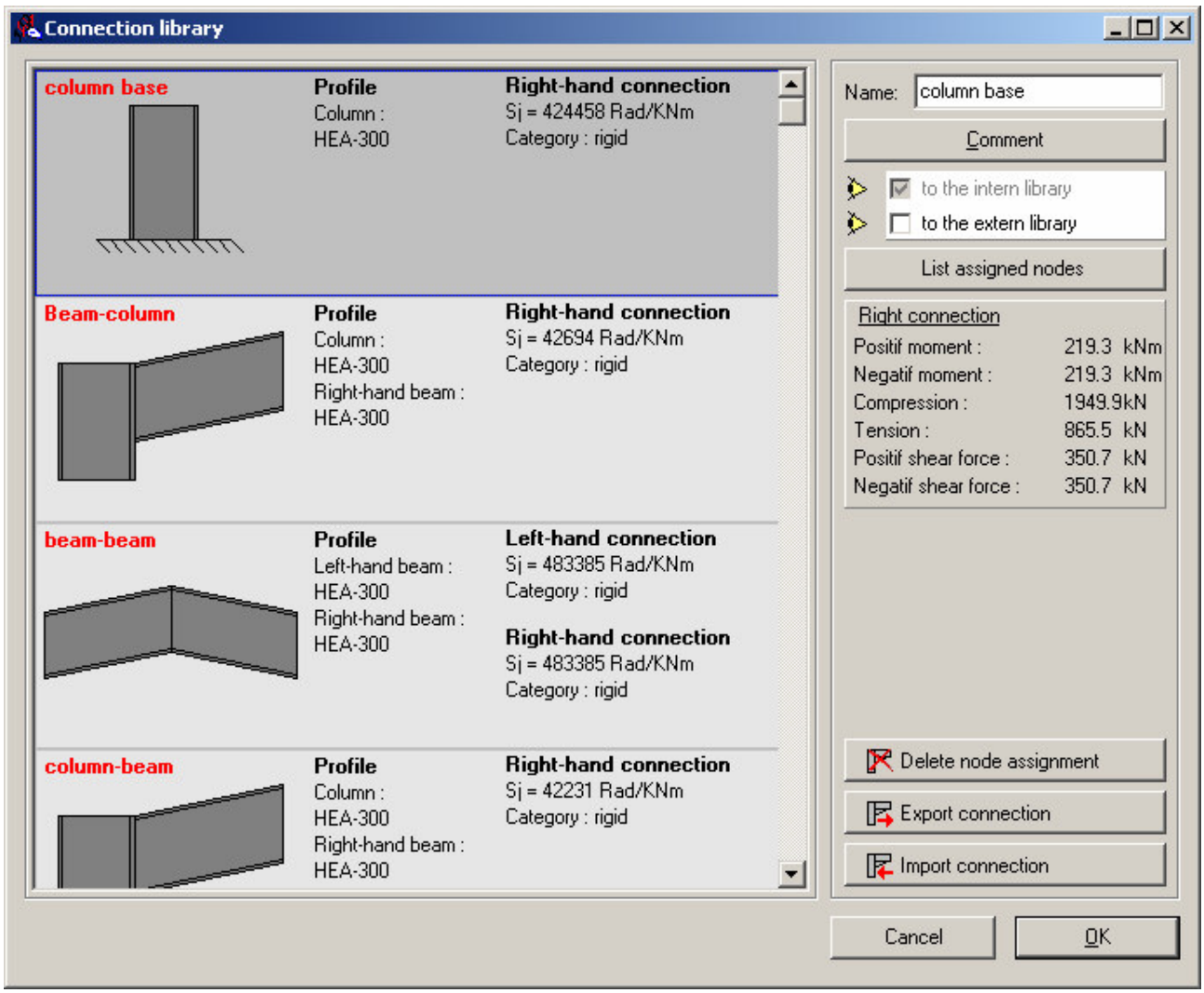
- case 3: no connection is available in the library which would fit to the selected node. The program will present an empty list when the icon is used.

Note: cases 1 and 2 are applicable only when one single node has been selected. If no node or multiple nodes have been selected, the user will find himself automatically in case 3.

Case 1 only shows the connection that is assigned to the selected node. The button 'Remove node assignment' at the bottom of the dialogue window allows to remove that connection from the selected node. However, this does NOT remove the connection from the internal and/or external library. At a later stage, this connection can still be assigned to another node.

If the user has selected a node to which no connection has been assigned yet (case 2), the dialogue will contain a button 'Assign connection', which enables the user to assign a connection (from the list shown at the left-hand side) to the selected node. Those connections can come from both the internal and the external library, provided that the buttons at the right-hand side are activated. A connection taken from the external library will automatically be added to the internal library, to ensure it will be saved along with the global analysis model.

Finally, the last case allows to consult and to manage connection library contents.



In all cases, detail information on connections accessed through the connection library is presented in the dialogue window shown above. In the left-hand side of this dialogue, a list is presented of all relevant available connections along with their name, type of connected cross-sections and stiffness characteristics. The button ‘Comment’ allows to add further information concerning the connection.

By selecting a specific connection from the list at the left-hand side, the resistance characteristics will be summarized by presenting the maximum values of the internal forces. To present the frame nodes to which this connection has been assigned, use the button ‘List assigned nodes’. The connection name can always be modified through the edit field at the top.

The two lines just below the ‘Comment’ button specify in which library the connection is stored. A ‘V’ sign indicates that the connection is saved in the appropriate library. A connection that has been saved in the internal library, can not be removed by switching of the ‘V’. If you still want to delete this connection, use the buttons at the lower part of this dialogue window. To visualize all connections present in the internal and external library, make

sure the icon is active. This button is independent of the selected connection at the left-hand side.

Finally, we'll explain the function of the three buttons at the bottom of the dialogue window.

The first one

allows to remove a connection from the library. All nodes to which this connection was assigned will loose the corresponding connection details and properties. Using the button 'Delete' will give the same result.

It's also possible to save and open connection properties without using any connection library through the buttons

and

. These connections are saved on the computer's hard disc with file extension '.cfr' and can be opened by PowerConnect without intervention of PowerFrame.

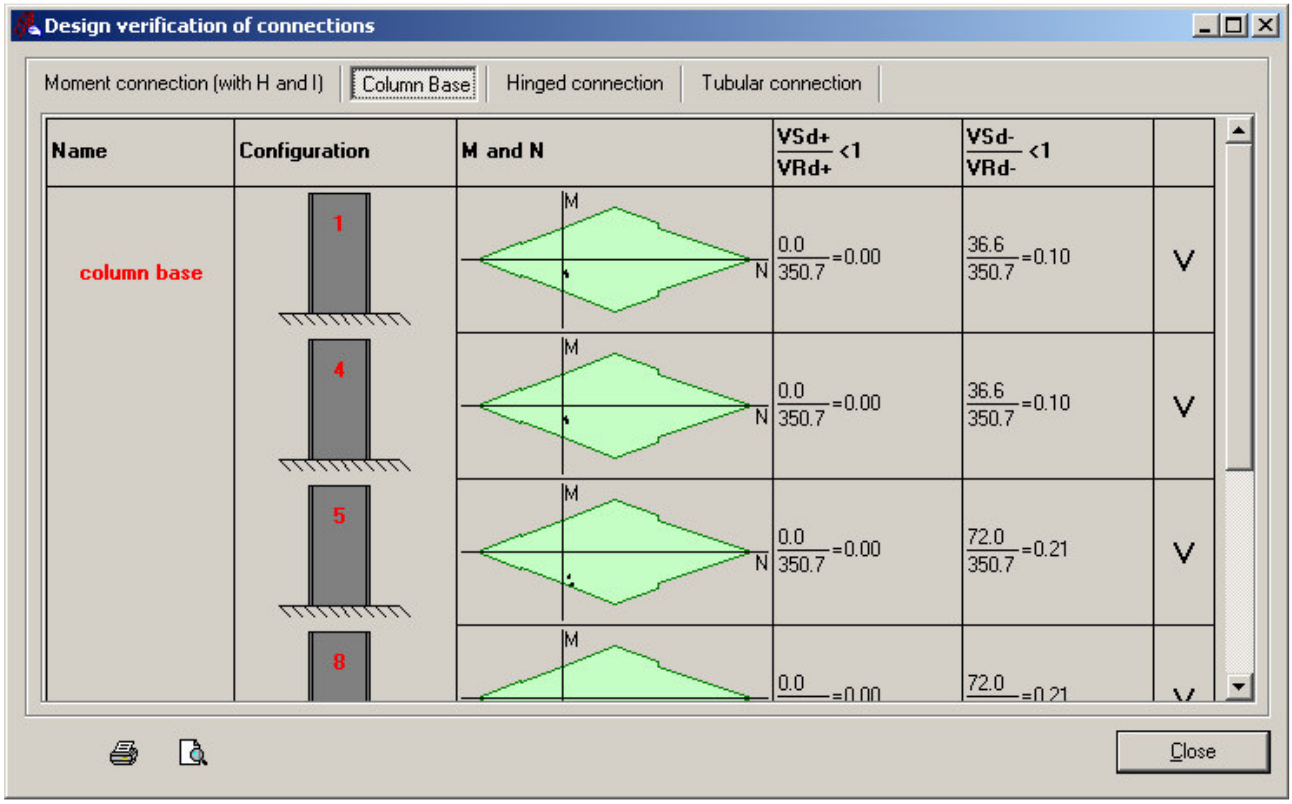
4.3 Verification of nodes & connections inside PowerFrame

Having assigned specific connections to several nodes in the global frame model, a new global analysis is required to account for the impact of connection properties on the elastic response of the structural frame. It can indeed be seen that, as a consequence of considering the actual stiffness of connections rather than considering them as either completely rigid or perfectly hinged, a redistribution of internal forces is possible. In principle, the user should then utilize those newly calculated internal forces to verify with PowerConnect if all connections are still sufficiently strong.

Rather than being forced to go through the detail connection analysis again in this final verification phase, PowerFrame offers the user the possibility to directly verify all connections in PowerFrame, thereby reducing considerably time and effort.

To launch the verification of connections within PowerFrame, use the icon. A window is then scheduled containing 4 tab pages. Each tab page corresponds to a specific connection type:

- moment connection (with H and I)
- column base (with H and I)
- hinged connection (with H and I)
- tubular connection



This window summarizes for each node the maximum internal forces and compares them to the strength characteristics of the corresponding connection, as previously transferred from PowerConnect to PowerFrame. In case this verification is unsatisfactory for a number of nodes, the verification results are printed in red.

The results of this verification can be printed using the icon at the bottom of the window. Alternatively, this verification list can also be included in an analysis report next to the other (global) analysis results (see 3.4.3.6).